Particles trajectory simulation in closed horizontal channels by Eulerian-Lagrangian approach

Maissam Vahidi[1], Farmarz Hormozi*
Chemical Engineering Department, Semnan University, Semnan, Iran.

Abstract
In this paper the trajectory of thin solid particles in air flow while passing a lab channel with a rectangular cross-section, have been simulated. Here we have used the Eulerian-Lagranjian method for modeling the two-phase flow, and the numerical solutions to governing equations have been made through a finite volume method. In Lagranjian calculations which are based on particle dynamics, the effects of any forces such as lifting, drag, and et cetera have been included. An second power upwind discreting approach has been used for the numerical solution of momentum equations, kinetic energy, dissipation rate and turbulence models; also the coupled calculations of pressure-velocity have been done through SIMPLEC algorithm. Comparison of the results obtained from lab data, indicate that the choose of parameters and turbulence model (Reynolds stress) is more suitable for dilute particle-fluid flow modeling inside closed channels.

Keywords: Two-phase modeling, Eulerian-Lagranjian approach, Particle Trajectory

1. Introduction
When solid particles are added to turbulent gas flows they change in the rate of gas turbulency, and eventually exert a deep effect on the behavior of median gas flows. However, if the solid volume fraction to be lower than10^{-5}, the momentum of particle transfer may be ignored and the movement of the gas and the particles may be modeled without taking into account the particle turbulence term. This calculation method has been called "one-way coupling", and the calculations of the two-phase flow in our study have been done according to this method. Up to now numerous methods have been proposed for the modeling of turbulence in continuous flows, in such a way that Banjerjee and Pan [1], Nadaoka [2], and Murai [3] have used direct numerical methods (DNS), and some others such as Matsumoto [4] and Liu Chunrong [5] have used the Large Eddies Simulation (LES) methods to model the turbulence flow patterns. This is while in recent years people such as Benny Kuan [6], Matheo Chiesa [7] have used Reynolds average time methods (RANS) to predict the turbulent behavior of continuous fluids. Though application of DNS methods in flow turbulence modeling has a good rate of accuracy, yet the increasing number of calculation cells complicate equation solutions and increase calculation costs in such methods. Application of LES methods is not recommended due to lack of predicting the details of turbulence behavior, specially in small particles affected by smaller eddies.

2. Test facility and flow conditions
In this study we have used the results calculated in a laboratory unit that was built by Summerfield and Kussin [8]. The said lab unit consists of a horizontal channel with a length, widths, and height of 6, 0.35, and 0.035 meters. As can be observed, the ratio of width to the height of the channel has been chosen in such a way that the assumption of two-dimensional calculations seems reasonable to a reliable degree. The top and bottom walls of the channel are made of stainless steel, while the side walls are made of glass for the purpose of laser photography. The velocity of air is controllable by a centrifuge fan, and has been taken as 19.7 m/s in this study. Glass spheres with an constant diameter

[1] . maissam_vahidi@yahoo.com

of 100 mm are released into the air flow field from a feeder. This keeps the solid/gas mass loading ratio well at 10% thus ensuring a dilute gas-solid flow regime inside the test section (figure 1).

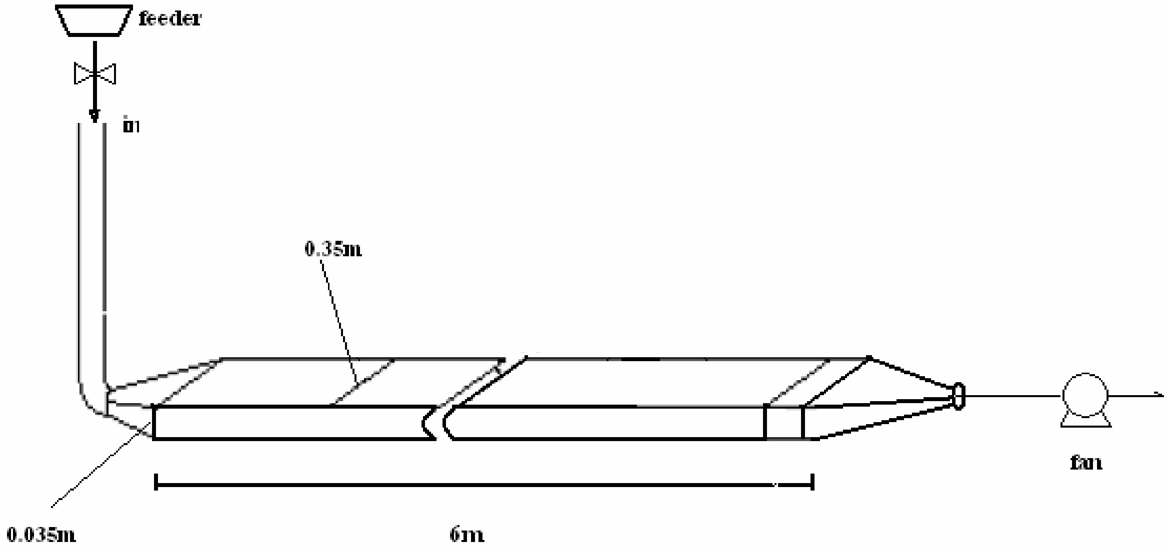


Figure 1: lab duct geometry by Summerfield and Kussin [8].

3. Theoretical background

Most studies conducted on the behavior and characteristics of flows have been based on phase particles continuity in calculation cells, which are known Euler-Euler modeling. However, studying the two-phase flow of fluid-particle in order to predict the particle trajectory requires the simultaneous solution of dynamic equations governing each separate particle. In this method (Eulerian-Lagrangian) first, the equations governing the continuous flow are solved and the obtained results are replaced in the Newtonian particle equations for every computational node, so that the coordinates of the particle in the course of time are compute.

3.1. gas flow

The governing equations of the gas flow for RSM are the two-dimensional continuity and Navier-Stokes equations. The Reynolds-averaged form of the conservation equations of mass(overall) and momentum for this incompressible fluid (air) can be written as:

$$\nabla.(r\,\overline{U}) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(r\,\overline{U}) + \nabla.(r\,\overline{U}\,\overline{U} + r\,\overline{U^{'}}\,\overline{U^{'}}) = -\nabla\overline{P} - \nabla.\overline{t} + rg \qquad (2)$$

Where the overbar indicates a time-averaged value. U' is the fluctuating velocity. Steady-state, isothermal gas flow properties and turbulence quantities are calculated numerically by solving a set of Reynolds averaged Navier-Stokes partial differential equations using a commercial CFD software fluent-6.2. In this work, Reynolds stress model (RSM) is used for prediction of gas turbulence behavior. Since the RSM accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than one-equation and two-equation models, it has greater potential to give accurate predictions for complex flow. The following differential Reynolds stress model (DRSM) as reported in Launder et al. [9] was utilized:

$$\frac{\partial}{\partial t}(r\overline{u_iu_j})+\frac{\partial}{\partial x_k}(rU_k\overline{u_iu_j})=\frac{\partial}{\partial x_k}(\frac{m_t}{s_k}\frac{\partial\overline{u_iu_j}}{\partial x_k})+\frac{\partial}{\partial x_k}[m\frac{\partial}{\partial x_k}(\overline{u_iu_j})]$$
$$-r(\overline{u_iu_k}\frac{\partial U_j}{\partial x_k}+\overline{u_ju_k}\frac{\partial U_i}{\partial x_k})+b\frac{m_t}{\mathrm{Pr}_t}(g_i\frac{\partial T}{\partial x_j}+g_j\frac{\partial T}{\partial x_i})+f_{ij,1}+f_{ij,2}+f_{ij}^w$$
$$+\frac{2}{3}d_{ij}(re+re\,2M_t^2)-2r\Omega_k(\overline{u_ju_m}\in_{ikm}+\overline{u_iu_m}\in_{jkm}) \quad (3)$$

Where U and u respectively represent mean and fluctuating velocities; and $\overline{u_iu_j}$ denotes the Reynolds stress tensor. The pressure-strain functions reported by Gibson ann Launder [10,11] as:

$$f_{ij,1}=-C_1r\frac{e}{k}\left[\overline{u_iu_j}-\frac{2}{3}d_{ij}k\right] \quad (4)$$

$$f_{ij,2}=-C_2\left[(p_{ij}+F_{ij}+G_{ij}-C_{ij})-\frac{2}{3}d_{ij}(P+G-C)\right] \quad (5)$$

$$f_{ij}^w=C_1'\frac{e}{k}(\overline{u_ku_m}n_kn_md_{ij}-\frac{3}{2}\overline{u_iu_k}n_jn_k-\frac{3}{2}\overline{u_ju_k}n_in_k)\frac{k^{3/2}}{C_led}$$
$$+C_2'(f_{km,2}n_kn_md_{ij}-\frac{3}{2}f_{ik,2}n_jn_k-\frac{3}{2}f_{ik,2}n_in_k)\frac{k^{3/2}}{C_led} \quad (6)$$

Where $C_1'=0.5$, $C_2'=0.3$, $C_1=1.8$, $C_2=0.6$, n_k is the x_k component of the normal to the wall, d is the normal distance to the wall, and $C_l=(C_m^{3/4}/K)$, where $C_m=0.09$ and $k=0.41$. Dissipation rate (e) and turbulent kinetic energy (k) can be written as below:

$$p\frac{De}{Dt}=\frac{\partial}{\partial x_j}\left[(m+\frac{m_t}{s_e})\frac{\partial e}{\partial x_j}\right]+C_{e1}\frac{1}{2}[P_{ii}+C_{e3}G_{ii}]\frac{e}{k}-C_{e2}r\frac{e^2}{k} \quad (7)$$

$$p\frac{Dk}{Dt}=\frac{\partial}{\partial x_j}\left[(m+\frac{m_t}{s_k})\frac{\partial k}{\partial x_j}\right]+\frac{1}{2}(p_{ii}+G_{ii})-re(1+2M_t^2) \quad (8)$$

Where $m_t=rC_m\frac{K^2}{e}$.

3.2. solid phase

Instantaneous positions and velocity of the dispersed phase are solved from a set of ordinary differential equations in the lagrangian domain:

$$\frac{du_P}{dt}=F_D+F_l+F_g+F_p \quad (9)$$

Where subscript P represents particle properties and subscripts D, l, g and p respectively denote force components arising from drag, shear-slip lift, gravity and flow pressure gradient.

3.2.1. Drag force
The drag force acted on a single particle is:

$$F_D = \frac{18\,m}{P_P D_P^2} \times \frac{C_D R_e}{24}(u - u_P) \tag{10}$$

Re is the relative Reynolds number, which is defined as

$$\mathrm{Re} = \frac{r D_P |u_P - u|}{m} \tag{11}$$

The drag coefficient, C_D, can be written as

$$C_D = \frac{24}{\mathrm{Re}}(1 + b_1 \mathrm{Re}^{b_2}) + \frac{b_3 \mathrm{Re}}{b_4 + \mathrm{Re}} \tag{12}$$

Where

$b_1=2.388-6.4581Ø+2.4486\,Ø^2$

$b_{2==}0.0964+0.5565\,Ø$

$b_3=4.905-13.8944\,Ø+18.42\,Ø^2-10.26\,Ø^3$

$b_4=1.4681+12.2584\,Ø-20.73\,Ø^2+15.8855\,Ø^3$

Which is taken from Haider and Levenspiel [59].

3.2.2. shear-slip lift force
Small particles in a shear field (such as high gradient velocity field near the wall) as shown in figure 2 experience a lift force perpendicular to the direction of flow. The shear lift originates from the inertia effects in the viscous flow around the particle and is fundamentally different from aerodynamic lift force. The modified expression for the shear lift obtained by Li and Ahmadi [12]:

$$F_x = \frac{2kn^{1/2} r d_{ij}}{r_p D_p (d_{lk} d_{kl})^{1/4}}(u - u_p) \tag{13}$$

Where k = 2.594 and d_{ij} is the deformation tensor.

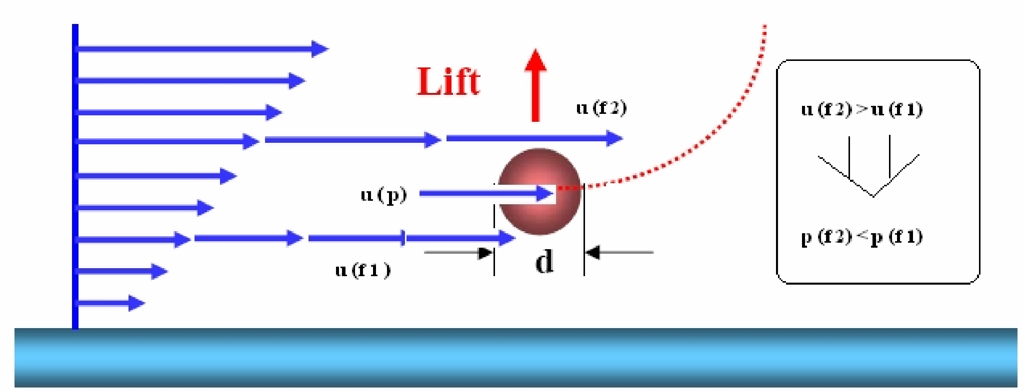


figure 2 : Schematics of a particle near a wall in a shear flow

3.3. Numerical procedure

The governing equations are solved by a finite volume method, where the calculation domain is divided into a finite number of non-overlapping control volumes. the conservation equations are integrated in space and time. This integration is performed using second power upwind discreting scheme in space. The SIMPLEC algorithm is used by Fluent to relate the velocity and pressure corrections to recast the continuity equation in terms of a pressure correction calculation. The two dimensional calculation domain is divided into 42000 (70×600) grid nodes, in two direction respectively. Grid structure is shown in figure 3.

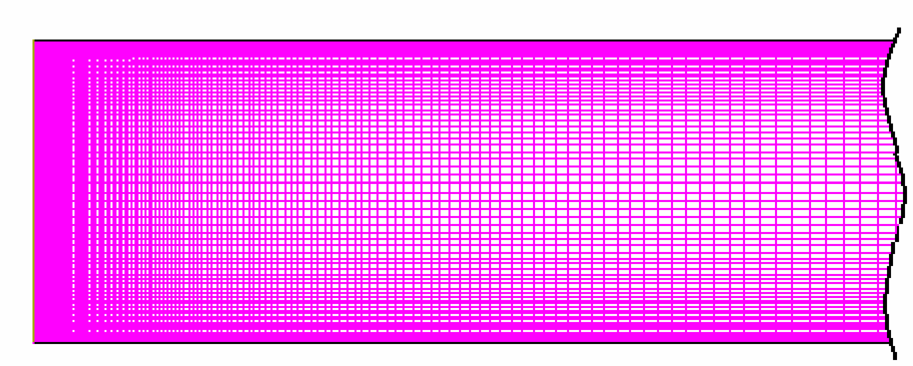
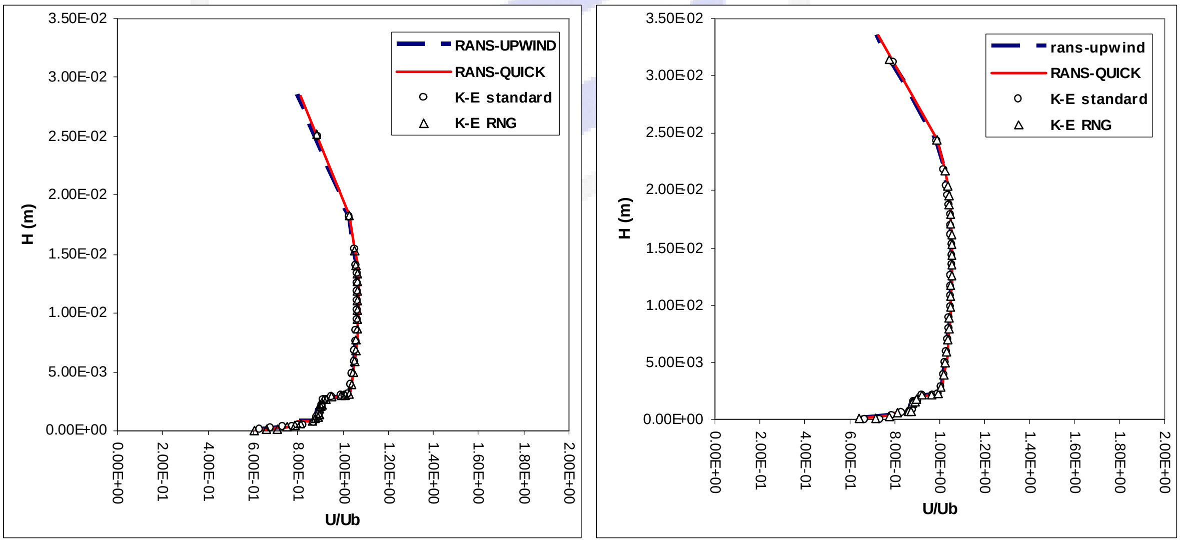


Figure 3: lab geometry and it's grid nodes

4. Results and Discussion

4.1. Comparison of two turbulent model effect on particle trajectory

Comparison of the results obtained from Reynolds Stress Model with $k-e$ models shows that there is no great difference between particle trajectory simulation results obtained from these models in dilute two phase flow wile passing a closed channel (figure 4).



A B

Figure 4 : Comparison of two turbulent model effect on velocity profile. A-section x=40H. B-section x=30H

4.2. Lift force effect

Particle velocity distribution results obtained from simulation with and without lift force effect at x=40H reported in figure 5.

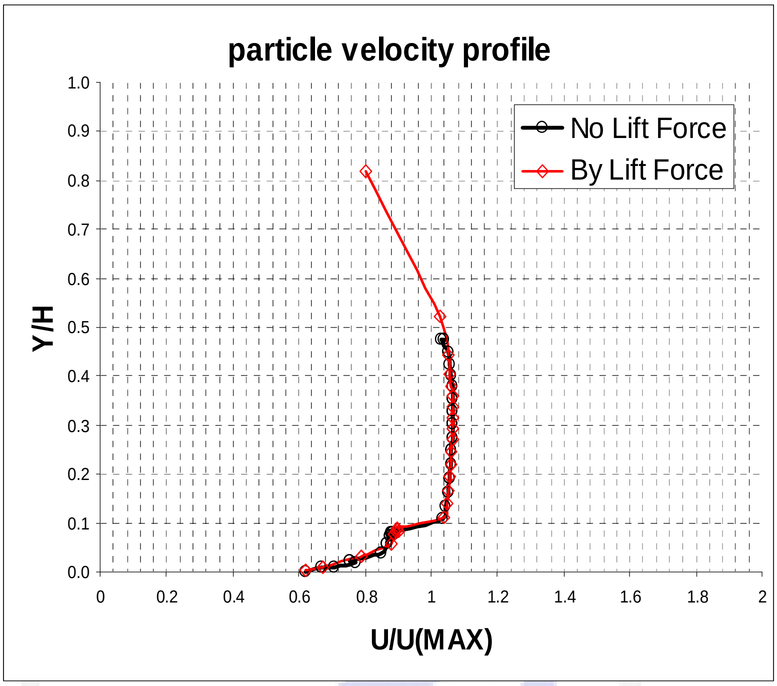


Figure 5: Velocity profile results obtained from simulation with and without lift force effect at x=40H

In 2003 Kuan conducted the above test in the Summerfield & Kussin lab channel, and presented its results in five sections (X/H =0, 10, 20, 30, 40). Therefore, the results of modeling and simulation of particle trajectory and velocity in these sections which were obtained in this study are presented in figure 6. Since the lack of expansion of the flow in upwind areas has caused a difference in the turbulence behavior the beginning and end of the channel, and given the fact that second calculation methods take into account the effects of upwind flows more from the first order methods, in this study the scheme of second Upwind have been used. Comparison of the results with the results presented by Kuan shows the desirable accuracy and quality of the calculations and the suitable choice of method and parameters. As seen in figure 6, predicting the arching spot of the curve at the lower part of the channel, indicating the effects of the continuous flow boundary layer on the movement of the particles, has been done correctly. In view of our investigations, it seems that the use of Reynolds Stress Model for modeling of the turbulence patterns in horizontal channels, and applying the effects of lift, drag, buoyancy and gravity forces on dispersed particles will provide an appropriate simulation of the particle trajectory .

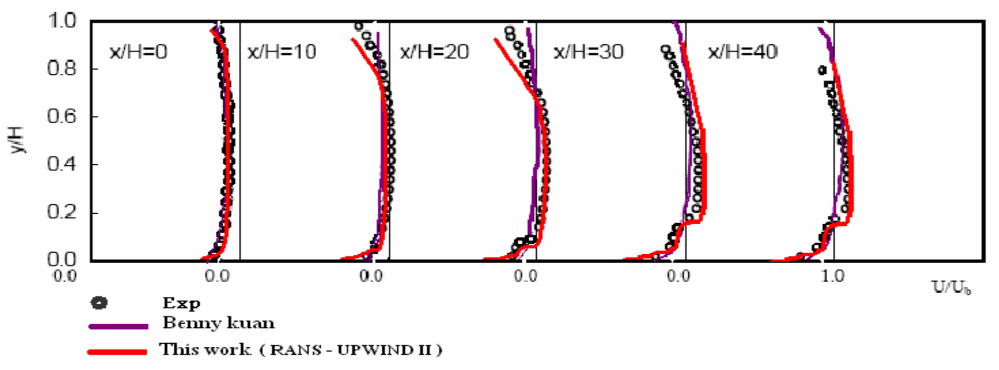


Figure 6 : Comparison of the results obtained from Reynolds Stress Model with lab results presented by Kuan

Nomenclature

$s_{\in} = 1, C_{\in 1} = 1.44, C_{\in 2} = 1.92$		m	Molecular viscosity
$P = 1/2(P_{kk})$	Stress production in kk direction	u_P	Particle velocity
$G = 1/2(G_{kk})$	Buoyancy in kk direction	u	Fluid velocity
$C = 1/2(C_{kk})$	Convection in kk direction	r	Fluid density
F_{ij}	Production by system rotation	r_P	Particle density
M_t	Much number	Ø	Shape factor

Conclusion

Comparison of the results obtained from lab data and this modeling results, indicate that the Reynolds stress turbulency moled with one way coupling method is more suitable for dilute particle-fluid flow modeling inside closed channels and Lift force is too effective for particles trajectory simulation .

References

1. Pan, Y .,Banerjee, S., "Numerical investigation of the effects of large particles on wall-turbulence" . Phys. Fluids 9, 3786-3807 (1997) .
2. Nadaoka, K., Nihei, Y., Yagi, H., "Grid-averaged Lagrangian LES model for multiphase turbulent flow", int. J. Multiphase Flow 25,1619-1643 (1999) .
3. Murai, Y., Matsumoto, Y., Song, X-Q., Yamamoto, F.," Numerical analysis of turbulence structures induced by bubble buoyancy" , JSME Int. J. Ser .B 43 (2), 180-187 (2000).
4. Matsumoto, Y., "Large eddy simulation of bubbly turbulent channel flow " , 3rd int. Conf. on Multiphase Flow , paper No.280 (1998).
5. Chunrong, K., Yincheng, G., "Mechanisms for particle clustering in upward gas-solid flow ",Chinese. J . Chem. Eng ., 14(2) 141-148 (2006) .
6. Kuan, B., Schwarz, M., "Numerical prediction of dilute particulate flows in horizontal and vertical ducts ", CSIRO Australia, December (2003).
7. Chiesa, M., Mathiesen, V., Melheim, J., Halvorsen, B., " Numerical simulation of particulate flow by Eulerian-Lagrangian and the Eulerian-Eulerian approach with application to a fluidized bed " , computers and chemical Eng ., 29, 291-304 (2005) .
8. Kussin, J., Sommerfeld, M ., "Experimental studies on particle behaviour and turbulence modulation in horizontal channel flow with different wall roughness " ,Experiments in fluids, 33, 143-155 (2002).
9. Lander, B., Reece, G., Rodi, W., " Progress in the Development of a Reynolds-Stress Turbulence Closure" , Fluid Mech.,68(3):537-566, (1975)
10. Lander, B., " Second-Moment Closure and its use in modeling turbulent industrial flows" , Int. j. for numerical methods in fluids, 9:963-985, (1989)
11. Lander, B., " Second-Moment Closure: Present… and Future?" Int. j. Heat fluid flow, 10(4):282-300, (1989)
12. Li, A., Ahmadi, G., " Dispersion and deposition of spherical particles from point sources in a turbulent channel flow " , Aerosol science and technology, 16:209-226, (1992)