HEFAT2010
7th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
19-21 July 2010
Antalya, Turkey

TURBULENCE MODULATION OF GAS-PARTICLE FLOW IN A VERTICAL CHANNEL USING EULERIAN-LAGRANGIAN APPROACH

Abkar M.[a], Mansoori Z.[b] and Saffar-Avval M.[a]*
*Author for correspondence
[a] Department of Mechanical Engineering,
[b] Energy Research Center,
Amirkabir University of Technology,
Tehran, Iran,
E-mail: mavval@aut.ac.ir

ABSTRACT

Tturbulent gas–solid flow in a vertical channel is investigated numerically using Eulerian–Lagrangian approach. Low Reynolds Number *k-l* model is used for studying the fluid phase. A new model is presented based on a source-term formulation, which can predict turbulent augmentation due to large particles and reduction of turbulence due to small particles in the core of the channel. The particle trajectories and velocities are determined by integrating the particle equations of motion. Particle-particle and particle-wall collisions are simulated based on deterministic approach and coupling terms representing the fluid-particle interactions are also taken into account. In this study, dilute or moderate suspensions are considered. The predicted fluid mean velocity and turbulent intensity profiles are in fine agreement with the available experimental data. Additional numerical results such as the eddy viscosity, the turbulent production and dissipation are also investigated for different values of loading ratio by means of a complete four-way coupling description. The results, comparing to integral length scale, show large particles augmenting turbulence in the core of the channel and small particle reducing turbulence. These effects are increased by inter-particle collision and increasing loading ratio.

INTRODUCTION

Turbulent gas-solid flows are frequently found in technical and industrial processes, such as chemical reactor, fluidized beds, dryers, waste treatment based on fluidized reactor, mixing devices, combustion of pulverized coal particles and more important in air pollution control. Such particle-laden flows, are affected by various parameters, such as loading ratio, particle characteristics, inter-particle collisions, particle-wall collisions and particle-turbulence interactions. One of the important issues in the development of modeling fluid-particle flows is the turbulence variation of the carrier phase which is responsible for the mixing, the particle dispersion and the heat transfer [1]. Several experiments have been performed on turbulence modulation in gas-solid flow [2-9]. The experimental data showed a general trend for small particles to attenuate turbulence and large particles to enhance turbulence. Gore and Crowe [10] evaluated the available experimental data on turbulence modulation effects and found the ratio of particle diameter to integral length scale of the fluid (d_p/l) to be a significant factor. When this ratio is less than 0.1, the data show that the turbulence level decreases with the addition of particles, whereas the level increases for d_p/l ratios greater than 0.1.

Several physical models have been proposed to explain the observed trends [11-16]. Various numerical models for multiphase turbulent flows are summarized in paper of Crowe et al. [17]. Kenning and Crowe [18] have proposed a model for turbulence modulation and the dissipation rate, based on the work done by particle drag and a length scale corresponding to the inter particle spacing, respectively. Crowe [1] derived and improved a detailed turbulence modulation model that has shown reasonable agreement results in with the limited available data.

As mentioned above, an accurate computational method for such flows in order to interpret and predict the experimental data is essential. In this study, the turbulent gas solid flow in a vertical channel is investigated using Eulerian–Lagrangian approach based on a four way coupling interaction for gas-solid flow. The numerical simulation is based on the experimental study performed by Tsuji et al. [9] for a vertical pipe. Inter-particle collision is treated by the deterministic method. In order to take into account turbulence modulation by particles, implementing the formulation by Crowe [1] which includes the extra-production term in the source term for the turbulent kinetic energy and the extra-dissipation term by defining hybrid length scale was opted. Thus, Low Reynolds Number *k-l* model is used for studying the fluid phase, the simulation result is compared with available experimental data. The influence of turbulence modulation on the turbulent kinetic energy, the eddy viscosity, production and dissipation of turbulence are studied. The results show that this approach can foretell both augmentation and reduction of turbulence due to large and small particles respectively.

NOMENCLATURE

A_p	[m^2]	surface area of particle
d_p	[m]	particle diameter
e	[-]	restitution coefficient
F_D	[N]	drag force
F_{LS}	[N]	slip-shear lift force
F_{LR}	[N]	rotational lift force
g	[ms^{-2}]	gravity acceleration
k	[m^{2}s^{-2}]	turbulent kinetic energy
L	[m]	initial dissipation length scale
l_h	[m]	hybrid dissipation length scale
m_p	[kg]	particle mass
M_r	[-]	solid loading ratio
p	[Nm^{-2}]	pressure
Re	[-]	channel flow Reynolds number
Re_p	[-]	particle Reynolds number
S_{pk}	[m^{2}s^{-3}]	kinetic energy source term
S_{pu}	[kgm^{-2}s^{-2}]	momentum source term
s_{pu}	[kgm^{-2}s^{-2}]	fluctuation of momentum source term
U	[ms^{-1}]	instantaneous velocity
U_c	[ms^{-1}]	fluid velocity at the centre of channel
u	[ms^{-1}]	velocity fluctuation
u^*	[ms^{-1}]	friction velocity
x	[m]	axial co-ordinate
X_p	[m]	particle location
y	[m]	distance from the wall

Special characters

ε	[m^2 s^{-3}]	Dissipation rate of turbulent kinetic energy
μ	[-]	Friction coefficient
δ	[m]	Channel half width
φ	[-]	Solid volume fraction
ν	[m2 s -1]	kinematic viscosity
ν_t	[m^2 s^{-1}]	eddy viscosity
ρ	[kgm^{-3}]	density
τ_p	[s]	particle dynamic relaxation time

Subscripts

f	Fluid
p	Particle
w	Wall
m	mean quantity across the channel
-	time average quantity
+	non-dimensional parameter

MODEL FORMULATION

The mathematical model of turbulent gas–solid flow in a vertical channel is developed based on Eulerian–Lagrangian approach. In hydrodynamic formulation the gas flow is assumed to be incompressible and hydrodynamically fully developed, convective terms being neglected and velocity profiles only altered due to fluid particle interactions). The solid phase is modeled by Lagrangian simulation and particles are solid, spherical, and with a constant diameter. The resulting equation system is closed by means of standard *k–l* model.

Gas Phase Equation

Momentum equation:

$$-\frac{(1-\varphi)}{\rho_f}\frac{d\bar{P}}{dx}+\frac{d}{dy}\left[\rho_f(1-\varphi)(\nu+\nu_t)\frac{d\bar{U}_f}{dy}\right]+\frac{1}{\rho_f}S_{pu}=0. \qquad (1)$$

Turbulent energy equation:

$$\frac{d}{dy}\left[(1-\varphi)\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{dk}{dy}\right]+(1-\varphi)\nu_t\left(\frac{d\bar{U}_f}{dy}\right)^2-(1-\varphi)\varepsilon + \frac{1}{\rho_f}S_{pk}=0. \qquad (2)$$

Where dissipation rate (ε) is defined as follows:

$$\varepsilon = C_{d1}\frac{k^{3/2}}{l_h}(1-e^{-A_\mu R_k})+C_{d2}\nu\frac{k}{{l_h}^2} \qquad (3)$$

Where l_h is a hybrid length scale which approaches the inter-particle spacing for inter-particle spacing less than the dissipation length scale, approximated as follow:

$$\frac{1}{l_h}=\frac{1}{\lambda}+\frac{1}{l} \qquad (4)$$

Where

$$\lambda = d_p\left[\left(\frac{\pi}{6\varphi}\right)^{1/3}-1\right] \qquad (5)$$

And

$$l=\delta\left(0.14-0.08\left(1-\frac{y}{\delta}\right)^2-0.06\left(1-\frac{y}{\delta}\right)^4\right) \qquad (6)$$

Where λ is the inter-particle spacing introduced by Crowe [1], l the initial dissipation length scale for a channel [19], δ the channel half width and y the transverse co-ordinate measured from the wall of the channel.

Closure equation:

$$\nu_t = C_\mu\sqrt{k}\,l\,(1-e^{-A_\mu R_k}) \qquad (7)$$

In which $\sigma_k = 1.$, $C_{d1} = 0.1664$, $C_{d2} = 0.32$, $A_\mu = 0.03$, $C_\mu = 0.55$ and $R_k = \frac{\sqrt{k}\,l}{\nu}$.

The coupling terms S_{pu} and S_{pk} as proposed by Crowe [1] are given by:

$$S_{pu_i} = \varphi\left\langle -\rho_p\left(\frac{dU_{p_i}}{dt}-g_i\right)\right\rangle \qquad (8)$$

and

$$S_{pk} = \left|S_{pu_i}\langle U_{f_i}-U_{p_i}\rangle\right| + \langle s_{pu_i}u_{p_i}\rangle \qquad (9)$$

Particle Phase Equation

In the Lagrangian approach, a large number of particles are tracked through the fluid field. The motion of each individual particle is subjected to Newton's equation of motion. The forces considered to act on the particle are the drag force, the gravity, the slip shear lift force, and the lift force resulting from particle rotation in order to calculate their instantaneous position, velocity, and rotational velocity:

$$\frac{d\vec{X}_p}{dt}=\vec{U}_p \qquad (10)$$

$$m_p\frac{d\overrightarrow{U_p}}{dt}=\overrightarrow{F_D}+\overrightarrow{F_g}+\overrightarrow{F_{LS}}+\overrightarrow{F_{LR}} \qquad (11)$$

$$I_p\frac{d\overrightarrow{\omega_p}}{dt}=\vec{T} \qquad (12)$$

In equation (15), $\overrightarrow{X_p}$ are the coordinates of the particle position, $\overrightarrow{U_p}$ are the velocity components. $\vec{T}$ is the torque acting on the particle, $\overrightarrow{\omega_p}$ is the angular velocity, I_p is the moment of inertia for a sphere. $\overrightarrow{F_D}$ is the drag force and is calculated from:

$$\vec{F}_D=\frac{3}{4}\frac{\rho_f m_p}{\rho_p D_p}c_D(\vec{U}_f-\vec{U}_p)\left|\vec{U}_f-\vec{U}_p\right| \qquad (13)$$

The drag coefficient c_D is calculated using the standard empirical correlations for a rigid sphere:

$$c_D = \begin{cases} \frac{24}{Re_p}\left(1 + 0.15Re_p^{0.687}\right), & Re_p \le 1000 \\ 0.44, & Re_p > 1000 \end{cases} \tag{14}$$

where $Re_p = \frac{\rho_f D_p |\vec{U}_f - \vec{U}_p|}{\mu_f}$ is the particle Reynolds number. $\vec{F}_{LS}$ represents the slip-shear lift force that is taken into account in the expression derived by Saffman [20], and extended for higher particle Reynolds numbers [21,22]:

$$\overrightarrow{F_{LS}} = 1.615 D_p^2 \left(\rho_f \mu_f\right)^{1/2} \left(\frac{1}{|\overrightarrow{\omega_f}|}\right)^{0.5} c_{LS} \left[\left(\vec{U}_f - \vec{U}_p\right) \times \overrightarrow{\omega_f}\right] \tag{15}$$

Where $\overrightarrow{\omega_f} = 0.5\nabla \times \vec{U}_f$ is the fluid rotation and c_{LS} represents the ratio of the extended lift force to the Saffman force:

$$c_{LS} = \begin{cases} \left(1 - 0.3314\beta^{1/2}\right) e^{\left(-\frac{Re_p}{10}\right)} + 0.3314\beta^{1/2} & Re_p \le 40 \\ 0.0524\left(\beta Re_p\right)^{1/2}, & Re_p > 40 \end{cases} \tag{16}$$

And $\beta = \frac{1}{2}\frac{Re_s}{Re_p}$ where Re_s is the Reynolds number of the shear flow:

$$Re_s = \frac{\rho_f D_p^2 |\overrightarrow{\omega_f}|}{\mu_f} \tag{17}$$

The slip-shear lift force is of importance in strong shear gradients. In the channel flow, this force is dominated near the wall due to strong shear gradients and has great influence on particle settlement (in horizontal channel) and particle concentration across the channel width.

The rotational lift force (F_{LR}) and and the torque (T) for a rotating sphere were derived by Rubinow and Keller [23] and extended by Crowe et al. [22]:

$$\vec{F}_{LR} = \frac{\rho_f}{2}\frac{\pi}{4} D_p^2 c_{LR} |\vec{U}_f - \vec{U}_p| \frac{\vec{\Omega} \times \left(\vec{U}_f - \vec{U}_p\right)}{|\vec{\Omega}|} \tag{18}$$

In which $\vec{\Omega} = \vec{\omega}_f - \vec{\omega}_p$. The lift coefficient ($c_{LR}$) computed following the proposal by Oesterlé and Bui Dinh [24] for $Re_R < 140$:

$$c_{LR} = 0.45 + \left(\frac{Re_R}{Re_p} - 0.45\right) exp\left(-0.05684 Re_R^{0.4} Re_p^{0.3}\right) \tag{19}$$

Where

$$Re_R = \frac{\rho_f D_p^2 |\vec{\Omega}|}{\mu_f} \tag{20}$$

Where Re_R is the Reynolds number of particle rotation. Particle rotation is mainly induced by wall-particle and particle-particle collisions.

For the torque acting on a rotating particle the expression of Rubinow and Keller [23] was extended to account for the relative motion between fluid and particle and higher Reynolds numbers:

$$\vec{T} = \frac{\rho_f}{2}\left(\frac{D_p}{2}\right)^5 c_R |\vec{\Omega}| \vec{\Omega} \tag{21}$$

where the coefficient of rotation is obtained from Rubinow and Keller [23] and direct numerical simulations of Dennis et al. [25] in the following way:

$$c_R = \begin{cases} \frac{64\pi}{Re_R} & Re_R \le 32 \\ \frac{12.9}{{Re_R}^{0.5}} + \frac{128.4}{Re_R} & 2 \le Re_R < 1000 \end{cases} \tag{22}$$

All of the equations in this section described in detail by Sommerfeld (2003). All these fluid–particle interactions (force and torque) involve the instantaneous velocity ($U = \bar{U} + u$) of the fluid at the particle location, which is predicted by means of a model as described here. The expressions of the fluctuational components of gas velocity are obtained by a modified Gaussian random field model proposed by Kraichnan [26] which was extended to nonhomogeneous flows by Li and Ahmadi [27].

$$u_f^+(X^+, t^+) = \sqrt{\frac{2}{M}} \left\{\sum_{n=1}^{M} U_1\left[\cos\left(K_n . X^+ + \omega_n t^+\right)\right]\right\} + \sqrt{\frac{2}{M}} \left\{\sum_{n=1}^{M} U_2\left[\sin\left(K_n . X^+ + \omega_n t^+\right)\right]\right\} \tag{23}$$

Where X^+ denotes the position vector and all quantities are non-dimensionalized with friction velocity u^* and kinematic viscosity.

$$u_{fi}^+ = \frac{u_{fi}}{u^*}, t^+ = \frac{t{u^*}^2}{\nu}, x_i^+ = \frac{x_i u^*}{\nu}, U_1 = \varsigma_n \times K_n, U_2 = \xi_n \times K_n \tag{24}$$

The component of vectors ς_n, ξ_n and frequencies ω_n are picked independently from a Gaussian distribution with a standard deviation of unity. Each component of K_n is also a Gaussian random number with a standard deviation of 1/2. In equation (29) M is the number of terms in the series. Here $M = 100$ as suggested by Li and Ahmadi [27]. This equation generates an isotropic homogeneous turbulence. In the case applying the equation to non-homogeneous flows, a scaling method is added. That is $u_{fi}^+ = u_f^+(X^+, t^+) e_i(X^+)$ in which $e_i(X^+)$ are the shape functions for the axial, vertical and transverse rms velocities [26,27].

NUMERICAL PROCEDURE

Equations are numerically solved using a finite-difference scheme. The time dependent computations are carried out for a turbulent symmetric fully-developed channel flow. An iterative scheme is applied in order to allow four-way coupling to be taken into account. In order to accurately handle the problem, it is necessary that the mesh width be small enough near the wall. A logarithmic scheme is used for the transverse co-ordinate. All the calculations are performed with meshes guaranteeing grid-independent results with 100 nodes in the axial direction and 100 nodes in the lateral direction. The equations of particle motion are calculated by the fourth-order Runge–Kutta scheme. The Lagrangian time step for solving the particle equations is chosen as described by Lain et al. [28]. The particles are initially distributed randomly across the flow domain to achieve a uniform initial concentration distribution. The initial particle velocity is chosen as 80% of gas mean velocity. The deterministic process is used to simulate interparticle collisions. If the distance between particle centers is less than the particle diameter then collisions are assumed to be occurred. Whenever particle–wall collision is detected, the after wall collision velocities are obtained. The collision model used in this study follows mainly the methodology by Wang and Mason [29]. Any particle leaving the calculation domain (the end of channel length) would be replaced by a new particle entering in with the same velocity. This is done to keep the number of particles constant in calculation domain. Additionally, let us mention that due to the strong grid tightening associated with the use of a low-Reynolds number model, the particle diameter may exceed the mesh width in the near wall region. In this case, the source terms are distributed over several cells according to the particle surface area pertaining to each cell. All the details concerning the numerical processing may be found in Mansoori et al. [30].

NUMERICAL RESULTS

Experimental data of Tsuji *et al.* [9] in the vertical pipe experiment is used to validate the dynamic characteristics. In their experiments, polystyrene spheres with a density of $\rho = 1020$ kg/m ^{3} and diameters in the range 0.2-1 mm were suspended in the air. The pipe diameter is 30.5 mm and the flow is fully developed. Unless otherwise stated, the collision parameters are set to $e_p = 0.8$, $e_w = 0.9$, $\mu_p = 0.4$ and $\mu_w = 0.2$. In this study 2 cases for different particle sizes are considered. The studied cases correspond to an air flow loaded with particles with diameter of 1420 μm and 243 μm. Figures 1 and 2 show the velocity profiles normalized with the aid of the gas velocity at the centerline in a vertical channel. It is observed that the mean velocity to be flatter due to the presence of particles. It is clear that this effect is increased with increasing loading ratio and particle size. Although there is no experimental data for these ranges of particle size in a vertical channel, but similar trend is observed in the experimental study of Tsuji *et al.* [9] in a vertical *pipe.*

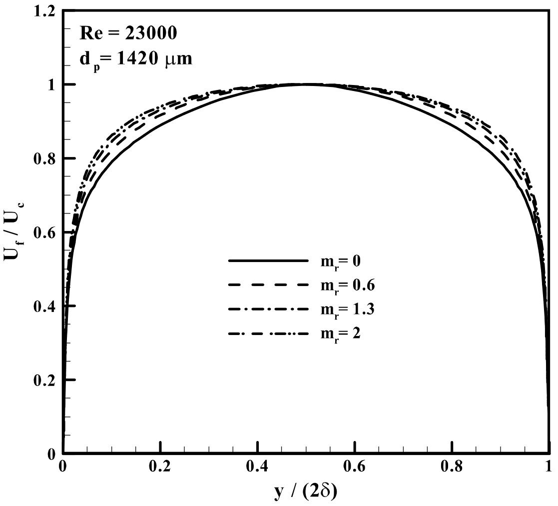

Figure 1 Mean fluid velocity$\left(\frac{\overline{U}_f}{U_c}\right)$

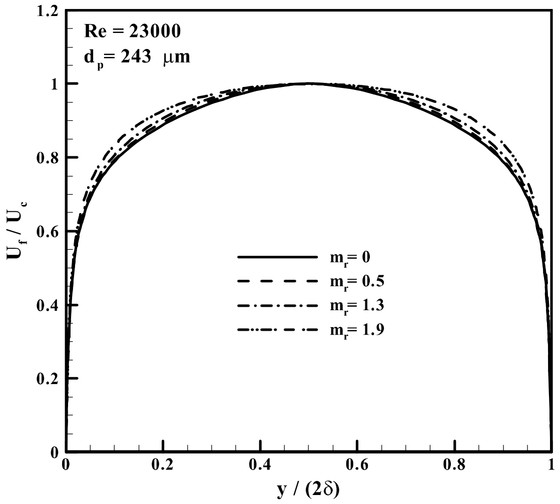

Figure 2 Mean fluid velocity$\left(\frac{\overline{U}_f}{U_c}\right)$

Figures 3 and 4 present the corresponding gas streamwise turbulence intensity (u_f) profile. The experimental data of Tsuji *et al.* [9] for RMS-axial fluctuating velocity is used for comparison. The model following the Crowe [1] formulation gives satisfactory results. With the presence of relatively large solid particles, the air turbulence intensities are increased in the core of the channel. As stated before, it is clear that the turbulence enhancement is due to large particles and the reduction is due to small particles. Figure 3 shows that turbulence increases in the core of the channel due to large particles. Similar results were reported in the experimental data of Tsuji *et al.* [9] for RMS-axial fluctuating velocity. In Figure 4 it is clear that turbulent intensity suppresses caused by small particles in low loading ratio and with increasing loading ratio it increases. A similar trend has been observed by Tsuji et al. [9]. The agreement is quite reasonable, in spite of the fact that model predictions are for a vertical channel while the data pertain to a pipe flow. It should be noted that the fluctuating velocity u$_f$ is calculated as: $u_f = \sqrt{\frac{2}{3}k}$.

The effects of particles and turbulence modulation on eddy viscosity are plotted in Figures 5 and 6. In clear fluid, eddy viscosity increases from zero at the wall and starts to decrease near the channel center. But it is observed that in gas-solid flow with large particles, with increasing loading ratio, eddy viscosity at the core of the channel starts to increase. This trend can be as a result of increasing the turbulent intensity in the center of channel because these two parameters directly relate to each other according to equation (7). In the presence of small particles, it is observed that eddy viscosity like turbulent intensity, with increasing loading ratio is initially decreased and then increased. It should be noted that, unfortunately there is no available data related to this parameter.

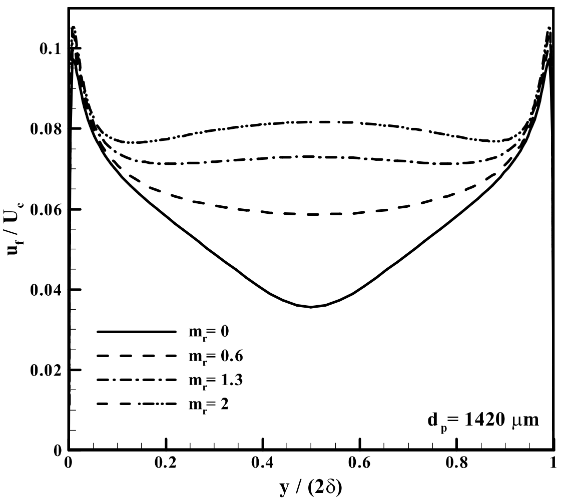

Figure 3 Air streamwise turbulent intensity

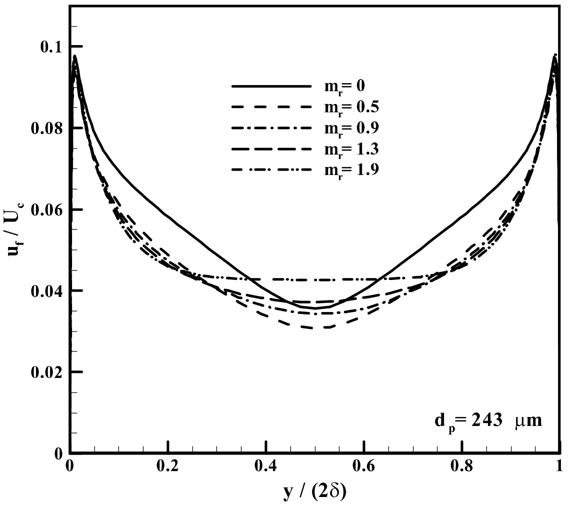

Figure 4 Air streamwise turbulent intensity

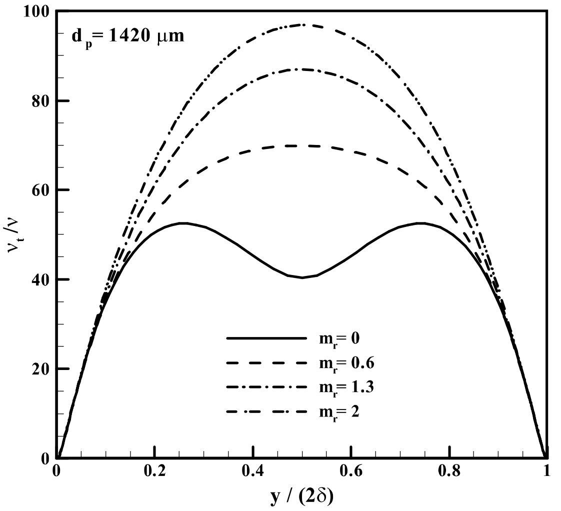

Figure 5 eddy viscosity

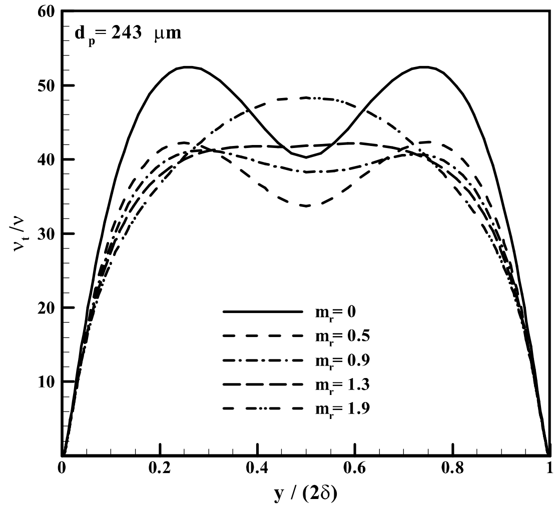

Figure 6 eddy viscosity

As stated before, in this model extra-production and extra dissipation due to particle-fluid interaction is considered. In Figures 7 and 8 the ratio of production $\left(\mathrm{P} = \nu_t \left(\frac{dU_f}{dy}\right)^2\right)$ to dissipation (ε) are plotted. These figures show that according to this model the ratio of production to dissipation due to particle-fluid interaction is decreased and this trend becomes more pronounced with increasing loading ratio and particle size. There is also no available data for comparison in this case.

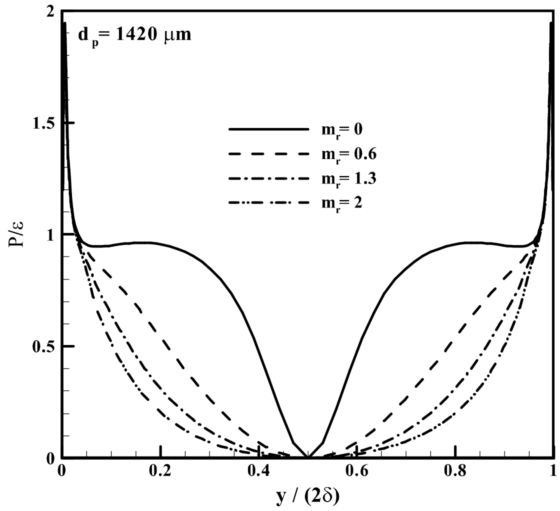

Figure 7 Production to dissipation ratio

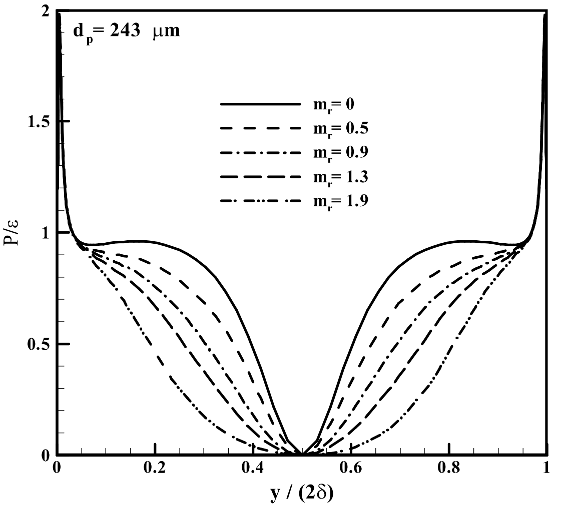

Figure 8 Production to dissipation ratio

CONCLUSIONS

Simulation of gas–solid turbulent flow in a vertical channel was performed by applying *k–l* turbulence modeling including coupling terms introduced by Crowe to simulate the fluid-particle interaction. Numerical calculations were carried out for different loading ratios. The results indicate that k-l model with coupling terms introduced by Crowe yield very interesting results. Up to now only the standard and the consistent approach were used in numerical investigation for particle-laden flow via Eulerian–Lagrangian approach. The standard approach is only able to predict attenuation of turbulence, whereas the consistent approach only contains mechanisms which can foretell enhancement of turbulence. But this new approach can predict both augmentation and reduction of turbulence due to large and small particles respectively. The effects of this source term on eddy viscosity, production and dissipation were also examined.

REFERENCES

[1] Crowe C.T., On models for turbulence modulation in fluid–particle flows, Int. J. Multiphase Flow, Vol. 26, 2000, pp. 719–727

[2] Kulick J.D., Fessler J.R., Eaton J.K., Particle response and turbulence modification in fully developed channel flow. J. Fluid Mech, Vol. 227, 1994, pp. 109-134

[3] Lee, S. L., Durst, F., 1982. On the motion of particles in turbulent duct flows. Int. J. Multiphase Flow 8, 125-146

[4] Parthasarathy R. N., Faeth G. M., Turbulence modulation in homogeneous dilute particle-laden flows. J. Fluid Mech, Vol. 220, 1990, pp. 485-537

[5] Savolainen K., Karvinen R., The effect of particles on gas turbulence in a vertical upward pipe flow. In: Third International Conference on Multiphase Flow, ICMF'98, Lyon, France, 1998.

[6] Schreck S., Kleis S. J., Modification of grid-generated turbulence by solid particles. *J. Fluid Mech. Vol.* 249, 1993, pp. 665-688

[7] Varaksin A.Yu., Kurosaki Y., Satoh I., Polezhaev Yu.V., Polyakov A.F., Experimental study of the direct influence of the small particles on the carrier air turbulence intensity for pipe flow. In: Third International Conference on Multiphase Flow, ICMF'98, Lyon, France, 1998.

[8] Tsuji Y., Morikawa Y., LFV measurements of an air–solid two-phase flow in a horizontal pipe. J. Fluid Mech. Vol. 120, 1982, pp. 385–409

[9] Tsuji Y., Morikawa Y., Shiomi H., LDV measurement of an air–solid two-phase flow in a vertical pipe, J. Fluid Mech. Vol. 139, 1984, pp. 417–434

[10] Gore R. A. and Crowe C, T., Effect of particle size on modulating turbulent intensity. Int.J. Multiphase Flow, Vol. 15, 1989, pp. 279-285

[11] Elghobashi S. E. and Abou-Arab T. W., A two-equation turbulence model for two-phase flows. *Phys. Fluids A, Vol.* 26, 1983, pp. 931-938.

[12] Elghobashi S. E., Truesdell G. C., On the two-way interaction between homogeneous turbulence and dispersed solid particles. I: Turbulence modification. *Physics of Fluids, Vol.5,* 1993, pp. 1790-1796

[13] Hwang G. J., Shen H. H., Fluctuating energy equations for turbulent fluid solid flows. *Int. J. Multiphase Flow, Vol.* 19, 1993, pp. 887-895

[14] Squires K. D. and Eaton J. K., Particle response and turbulence modification in isotropic turbulence. *Phys. Fluids* A, Vol. 2, 1990, pp. 1191-1203.

[15] Yarin L. P. and Hetsroni G., Turbulence intensity in dilute two-phase flows--3. Int. J.Multiphase Flow, Vol. 20, 1994, pp. 27-44

[16] Yuan Z., Michaelides E. E., Turbulence modulation in particulate flows a theoretical approach. *Int. J. Multiphase Flow, Vol.*18, 1992, pp. 779 785.

[17] Crowe C. T., Troutt T. R., Chung J. N., Numerical models for two-phase turbulent flows. Annual Rev. Fluid Mech. Vol. 28, 1996, pp. 1143

[18] Kenning V.M., Crowe C.T., On the effect of particles on carrier phase turbulence in gas-particle flows. Int. J. Multiphase Flow. Vol. 23, 1997, pp. 403-408

[19] Schlichting H., Grenzschicht-Theorie. Verlag G. Braun, Karlsruhe, 1982.

[20] Saffman P.G., The lift on a small sphere in a shear flow. J. Fluid Mech. Vol. 22, 1965, pp. 385–400

[21] Mei R., An approximate expression for the shear lift force on a spherical particle at finite Reynolds number. Int. J. Multiphase Flow, Vol. 18, 1992, pp.145–147

[22] Crowe C.T., Sommerfeld M., Tsuji Y., Fundamentals of gas–particle and gas–droplet flows. CRC Press, 1998

[23] Rubinow S.I., Keller J.B., The transverse force on spinning sphere moving in a viscous fluid. J. Fluid Mech. Vol. 11, 1961, pp. 447–459

[24] Oesterlé B., Bui Dinh T., Experiments on the lift of a spinning sphere in a range of intermediate Reynolds numbers. Exp. Fluids, Vol. 25, 1998, pp. 16–22

[25] Dennis S.C.R., Singh S.N., Ingham D.B., The steady flow due to a rotating sphere at low and moderate Reynolds numbers. J. Fluid Mech. Vol. 101, 1980, pp. 257–279

[26] Kraichnan H., Diffusion by random velocity field, Phys. Fluids, Vol. 11, 1970, pp. 22–31.

[27] Li A., Ahmadi G., Computer simulation of deposition of aerosols in a turbulent channel flow with rough walls, Aerosol. Sci. Technol. Vol. 18, 1993, pp. 11–24

[28] Lain S., Broder D., Sommerfeld M., Experimental and numerical studies of the hydrodynamics in a bubble column, Chem. Engrg. Sci. Vol. 54, 1999, pp. 4913–4920

[29] Wang Y., and Mason M. T., Two-Dimensional Rigid-Body Collisions With Friction, ASME J. Appl Mech., Vol. 59, pp. 536–642

[30] Mansoori Z., Saffar-Avval M., Basirat Tabrizi H., Ahmadi G., Modeling of heat transfer in turbulent gas–solid flow, Int. J. Heat Mass Transfer, Vol. 45, 2002, pp.1173–1184