International Journal of Science and Research (IJSR)
ISSN: 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2018): 7.426

Effect of Friction Reynolds Number on Turbulence Modulations in Liquid Flow with Presence of Dust Particles

Asaduzzaman[1], Dr. Md. Lutfor Rahman[2]

[1]North Bengal International University, Faculty of Science and Engineering, Dainik Barta Complex, Bangabandhu Chottar (Alupatti), Rajshahi, Bangladesh

[2]Department of Mathematics, University of Rajshahi, Rajshahi-6205, Bangladesh

Abstract: ***This Turbulence modulation, namely the effect of friction Reynolds number of dispersed dust particles on turbulence intensity, has attracted a lot of attention and been a long-term controversy in the multiphase flow community. The purpose of this paper is to analyze the behavior of dust particles industrial processes and final product quality. The turbulence modulation plays an important role in particles motion and the re-entrainment influence dust particles collection. Studies on dust particle distributions and interactions between the dust particles and the liquid turbulence are extremely significant and can help to improve efficiency of industrial processes and final product quality. In this paper, the dust particle distribution and the dust particle turbulence interaction in the solid-liquid flow were investigated in detail by a numerical method. The presence of dust particles in a solid-liquid two phase flow entails some relevant changes in the mean velocity field. The results are compared to the solid-liquid two phase flow in order to investigate the effect of friction Reynolds number with presence of dust particles on the turbulence modulation.***

Keywords: DNS, Dust particle, Turbulence modulation, Solid-liquid flows, Euler-Lagrange model

1. Introduction

Many studies have shown that certain additives can specially modify the turbulence of fluids. One of the most interesting problems of fluid dynamics is the solution of turbulent flow with presence of dust particle in the main stream. This type of flow can be found in many industrial applications, example-in the chemical industry, coal-fired power plants in the Transportation of coal dust to the boiler, in the prediction of the air pollution and many others. The studies showed that the effect of the dust particles in the liquid turbulence is extremely complicated and depends on many factors such as the dust size and shape, the void fraction, the gas and liquid velocity, the flow direction of the liquid etc.For a gas flow laden with dust particles, the studies showed that, for a low mass loading, the micro-particles increase the gas velocity and reduce the turbulent intensities in the wall-normal and span- wise direction. When the mass loading is high, the addition of the micro-particles greatly decreases the velocity, the turbulent intensity and the Reynolds stress of the gas. The micro-particle influence on the gas turbulence is also associated with many factors, including the particle size and density, the void fraction, the mass loading etc.It can be seen from the above review that there are some similarities and differences of the turbulence modulation caused by the different additives of micro-size. Particle setting and sedimentation is central to all disperse two phase flows in which the particle density is larger than the density of the carrier fluid. These flows include a variety of natural multiphase systems as well as engineering applications. Examples-are dust particle transport in the atmosphere, particle sedimentation in river beds or turbo inhaler devices for medical purpose. For the insoluble additives, there are three common numerical methods, including the Euler-Euler method, the Euler-Lagrange model and direct numerical simulations (Loth-2000, Lain, etal-2002).compared with gas-solid and gas-liquid two –phase flows, the related reports on the influence of solid particles on liquid turbulence are relatively few, though solid particles can also greatly modify the liquid turbulence. Solid-liquid two-phase flows are frequently observed in many industrial processes and natural phenomena. For example –for a complex material based on chopped fibers the distribution and orientation of the fibers at the liquid-phase stages, which are closely related to the liquid turbulence structure directly control the product quality. In view of the above analysis. We investigated in detail the effect of dust particles on liquid turbulent with an Euler-Lagrange model.

2. Numerical Simulation

The flow domain and the coordinate system are shown in figure (1), in which the x, y and z axes correspond to the stream wise and span wise wall-normal directions respectively. The flow domain size is 18h 3h 6h (=3600 600 1200 wall units).Here, h is the channel half-width; the liquid phase is considered to be an incompressible Newtonian fluid. Thermo-physical properties of water at room temperature were adopted. The particle parameters are listed in table (1). Where ρp ,dp, M, a0 and St denote the particle density, the particle diameter, the total number of dust particles , the average volume fraction, the average mass loading and the stokes number respectively. Periodic boundary conditions are adopted in both the stream wise and span wise directions and the non-slip condition is imposed at the right and left walls for liquid phase. Uniform grids are used in the stream wise and span wise directions. Non-uniform grids are generated by a hyperbolic tangent stretching function are imposed in the wall-normal direction. Our calculations based on the friction Reynolds number

R=240, which is calculated from the wall friction velocity and the channel half width. Initial velocities of the dust particles are assigned to zero.

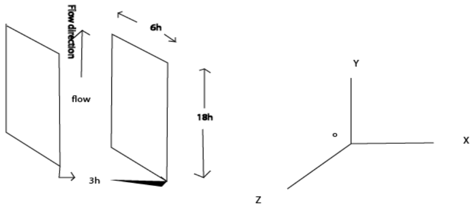

Figure 1: Flow geometry and coordinate system.

Table1: Parameters of the particles.

Parameters	$\frac{\rho_p}{\rho_f}$	$\frac{d_p}{h}$	M	α_0	$\emptyset$	S_t
Specific value	1.0	0.011	19200	1.3x10^{-4}	1.3 x10^{-4}	0.15

Direct numerical simulations have been executed in the plane-channel domain in **figure 2** with dimensions, grid points, resolutions and Reynolds numbers as reported in **table2**

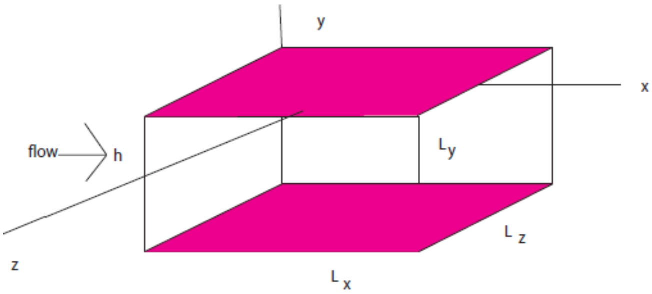

Figure 2: Computing-domain scheme

As concerns the calculation of the kolmogorov micro scales, they have been evaluated by estimating the average rate of dissipation of turbulent kinetic energy per unit mass (ε).

Table 2: Characteristic parameters of the numerical simulations

Quantities	Re_τ=200	Re_τ=400	Re_τ=600
L_x	4πh	4πh	4πh
L_y	2h	2h	2h
L_z	2πh	2πh	2πh
L_x^+	2513	5026	7540
L_y^+	400	800	1200
L_z^+	1256	2513	3770
N_x	256	343	512
N_y	181	321	451
N_z	256	343	512

This method was first introduced by Bake-well and Lumley, where in the case of the plane channel, one has

$$\varepsilon \cong \frac{2L_x L_z \tau_w u_b}{2\rho h L_x L_z} = u_\tau^2 u_b$$

The calculation has been executed on a specially assembled hybrid multi-core/many core computing architecture. A grid of $[N_x, N_y, N_z] = [128,256,128]$ on a domain size of $[L_x, L_y, L_z] = [2\pi h, 2\pi h, h]$ is used for the carrier phase calculations. The blank Reynolds number of the flow, defined as $Re_b = u_0 {}^h/_{\gamma_f}$, is set to 8000, which leads to a friction Reynolds number of $Re_\tau \approx 120$, where $Re_\tau = u_\tau {}^h/_{2\gamma_f}$,is based on the standard friction velocity u_τ. Both the stream-wise and span-wise domain distances are too small to avoid artificial confinement of the large-scale coquette structures which form in this flow, but for the purposes of this study this effect does not preclude a study of the effects of poly disparity on dust particle stresses. Solid-liquid dust particles were introduced in the gas stream. Their location X_p was updated according to $\frac{dx_p}{dt} = V_p$; the evolution of the particle velocity V_p obeyed Newton's second law with the particle feeling stokes drag and gravity

$$\frac{dv_p}{dt} = \frac{u_{in}}{S_t}(u - v_p) + g$$

Where u is the gas velocity at the location of the dust particles and g is the gravitational acceleration vector and S_{tk} is the stokes number (defined as $S_{tk} = \frac{\rho_p d_p^2 u_i}{\rho g 18 \gamma d}$) , dust particle sizes are of the order of a few microns, slip velocities of the order of a few meters per second at most and air under atmospheric conditions is the working fluid. This means that dust particles Reynolds numbers not exceeding unity. In the simulations, we achieve solid-liquid two phase flow by feeding back the force that the gas exerts on the dust particles as a source term to the filtered Navier-Stokes equations. Even when we consider only modest dust particle mass loading levels (of say, 0.01), the total number of dust particles inside the cyclone becomes extremely large and tracking the motion of all these dust particles is computationally unaffordable. In the simulations, the dust particles did not interact with one another.i.e. A dust particle does not undergo collisions with other dust particles. Based on the space averaged solid-liquids volume fraction that is of the order of 10^{-5} this is a fair assumption. However, since the bigger dust particles accumulate at the outer wall, locally the solid-liquids volume fraction becomes one to two orders of magnitude higher and we enter a region where dust particle-dust particle collisions may become important. However, we do not expect these near wall regions to significantly alter the turbulence. In the simulations, the default settings for dust particle-wall collisions are such that collisions are elastic and frictionless (smooth walls). We have checked the impact of these assumptions by performing a simulation with rough walls that we mimic with diffusive reflections of dust particles, and a restitution coefficient of 0.9.We assume the Reynolds number of dust particle Re_b based , on the angular velocity is negligible compared to the dust particle Reynolds number Re_p or Re_τ based on the translational velocity. This assumption is found to be valid in some circumstances with small shear force difference and velocity difference acting on the dust particle by fully resolved simulation of dust particle sedimentation near a two parallel walls.

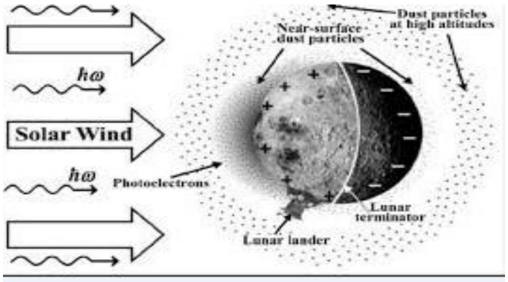

Figure 3: Turbulence modulation based on the dust particle Reynolds number.

3. Fluid Field

The Navies-Stokes equation of motion and continuity for viscous incompressible dusty fluid DNS velocity field of the liquid calculated is given by

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \delta_{1i} + \frac{1}{Re_\tau}\nabla^2 u_i - \overline{f_i} \tag{2}$$

In presence of dust particles the equation of motion are given by-

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \delta_{1i} + \frac{1}{Re_\tau}\nabla^2 u_i - \overline{f_i} + \frac{KN}{\rho}(v_i - u_i) \tag{3}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \delta_{1i} + \nu\nabla^2 u_i - \overline{f_i} + f_c(v_i - u_i) \tag{4}$$

Where, $\nu = \frac{1}{Re_\tau}$ friction Reynolds number on tensor.

$f_c = \frac{KN}{\rho}$ dimension of frequency

K=the stokes's resistance coefficient which for spherical particle of radius r is $6\pi\mu r$.

N=constant number of density of dust particle.

ρ= the density of the material in the dust particle.

δ_{1i}=average pressure gradient

$\overline{f_i}$=Local average interfacial force

v_i =solid particle of dust

u_i=fluid particle of dust

In the present simulations $\overline{f_i}$ is evaluated by

$$\overline{f_i} = \frac{\rho_i v_i}{\rho v}\sum_{j=1}^{N}(a_i)_j \tag{5}$$

Again, the present simulations of dust particle f is evaluated by

$$f = f_c \sum_{i=1}^{n}(v_i - u_i) \tag{6}$$

Now from equation (1) and (4) using fractional method begins from a time discretization of the Navier-Stokes equations. First, the second-order Adams-Bashforth for the convective terms and trapezoidal for the diffusive terms. This results is

$$\frac{u_i^{n+1} - u_i^n}{\Delta t} + \left[\frac{3}{2}(u_i^n.\nabla)u_i^n - \frac{1}{2}(u_i^{n-1}.\nabla)u_i^{n-1}\right] = -\nabla p^{n+1} + \frac{1}{2}\frac{1}{R_{e\tau}}\nabla^2(u_i^{n+1} + u_i^n) \tag{7}$$

$$\text{And} \quad \nabla.u_i^{n+1} = 0 \tag{8}$$

The idea is to approximate of equation (7) by calculating a tentative velocity u_i^+,using the momentum equations without the pressure. Mathematically this looks likes a simple time splitting of (7) is

$$\frac{u_i^+ - u_i^n}{\Delta t} + \left[\frac{3}{2}(u_i^n.\nabla)u^n - \frac{1}{2}(u_i^{n-1}.\nabla)u_i^{n-1}\right] = \frac{1}{2}\frac{1}{R_{e\tau}}\nabla^2(u_i^+ + u_i^n) \tag{9}$$

$$\frac{u_i^{n+1} - u_i^+}{\Delta t} = -\nabla\rho^{n+1} \tag{10}$$

The pressure in (10) is found by taking the divergence of equation (10) and invoking the incompressibility condition of equation (8).This results in the Poisson equation for the pressure

$$(\nabla.\nabla)p^{n+1} = \frac{1}{\Delta t}\nabla.u_i^+$$

$$\nabla^2 p^{n+1} = \frac{1}{\Delta t}\nabla.u_i^+ \tag{11}$$

At-every time-step, preceding equations (9),(10) and (11) are solved and Rearranging both the equations of their order gives

$$\frac{1}{\Delta t}\left[f - \frac{\Delta t}{2R_{e\tau}}\nabla^2\right]u_i^+ = \frac{1}{\Delta t}\left[f + \frac{\Delta t}{2R_{e\tau}}\nabla^2\right]u_i^n - \left[\frac{3}{2}(u_i^n.\nabla)u_i^n - \frac{1}{2}(u_i^{n-1}.\nabla)u_i^{n-1}\right] \tag{12}$$

$$u_i^{n+1} = u_i^+ - \Delta t.\nabla p^{n+1} \tag{13}$$

$$\Delta t(\nabla.\nabla)p^{n+1} = \nabla.u_i^+$$

$$\Delta t.\nabla^2 p^{n+1} = \nabla.u_i^+ \tag{14}$$

The divergence from is protective for the finite difference schemes when a staggered grid is used. It is clear that the fractional method is simply the continuous analog of the viscous incompressible dusty

Fluid DNS velocity decomposition. This analysis is reinforced by the fact that the error term found by adding (8) and (9) together and comparing to (6) is $(\frac{\Delta t}{2Re})\nabla^2\nabla p^{n+1}$which is also just the continuous version of what was found for the dusty fluid DNS velocity decomposition. The spatial derivatives are discredited by a second-order central difference scheme. For time integration, the Adams-Bashforth scheme is used for all the terms except that the implicit method is used for the pressure term. The Poisson equation for pressure is solved with the multi-grid method. Traditionally, it is only at this point in the fractional-method that equation (11),(12) and (13) are spatially discretized.The subtle differences between the traditional fractional-method and the dusty fluid DNS decomposition are due to the point at which spatial discretization and boundary conditions are implemented. By implementing boundary conditions before any splitting or decomposition takes place, the dusty fluid DNS factorization does not require any boundary conditions on intermediate variables or on the pressure. Additionally it should be pointed out that the pseudo-spectral method is often preferred because a higher numerical accuracy can be obtained for a given grid size using the pseudo-spectral method than the finite difference method. Thus, many researchers (such as Kim et al.,1987;Moster et al., 1999;Li et al.,2001; etc) have preferred to use the pseudo-spectral method to discretize the spatial terms of the governing equation in DNS.Attempts to improve the time order of accuracy of the fractional-method through improved boundary conditions on intermediate variables and pressure appear to be misted. The dusty fluid DNS decomposition shows that boundary conditions are not an issue and that first-order accuracy in time is a fundamental result of the method itself.

4. Motion of Particle on Reynolds Number

The motion of particles is described by Lagrangian equation of motion for each particle. The only force considered, here is drag force. Because of high concentration of particles we consider the influences of particles on the fluid, which is assumed to be the stokes drag with an empirical Reynolds number correction.

$$V_i = \frac{dx_i}{dt} \tag{15}$$

$$\frac{dv_i}{dt} = \frac{1+0.15Re^{0.687}}{\tau} f_c(U_i - V_i) \tag{16}$$

Where, is the velocity of the ith particle in the direction direction $\tau = \frac{\rho}{18}$ is particle relaxation time. $R_e = f_c d|U_i - V_i|$ Is the particle Reynolds number, the particle diameter and density are given by d and respectively, f $= f_c(U_i - V_i)$ Represent the dust particle of the fluid.

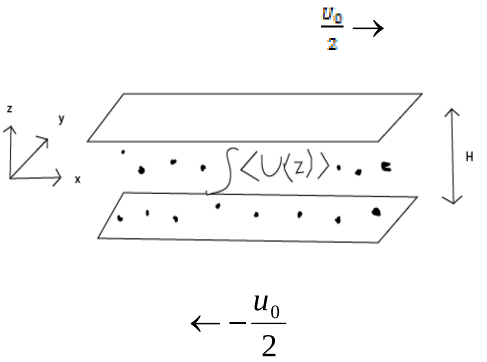

Figure 4: Simulation domain schematic. Plate velocity difference is u_0 and plate separation is H.

Turbulent planer Couette flow developed between two parallel plates moving with equal and opposite velocities. Where the velocity difference and H is the distance between the two parallel walls. See in **figure 4** .The opposite velocities and distance of two parallel walls are related to the dust particle of the bulk Reynolds number of the flow is defined as $\mathrm{Re}_\tau = u_0 \frac{H}{\gamma f}$, which leads to a friction Reynolds number of $\mathrm{Re}_\tau \approx 120$ .Furthermore turbulent statistics such as the Reynolds stress are relatively insensitive to this confinement. Turbulent Couette flow is chosen because the total stress is uniform with height and the presence of the large structures provides an additional time and length scale with which the dust particles can interact in addition to the typical near wall streaks and vortices found in turbulent channel flow. The simulation of dust particles-laden turbulent channel flow was performed. The simulated flow field is a fully developed turbulent flow between two parallel walls. Hence the flow is homogeneous both in the stream-wise and span-wise directions and the statistics are dependent only upon the distance from the two parallel walls. The simulations were performed for high Reynolds number ($Re_\tau = 400$). The concentration was chosen sufficiently high in order to get effect on the solid-liquid two phase flow with presence of dust particles.Turbulenence should be modulated by particles with presence of dust particles are supposed to be spheres.

5. Results and Discussion

In this section are introduced results and discussion of simulations of dust particle laden channel flow. First was done simulation without dust particles and the data from this simulation was used for comparison dust particle laden flows in order to investigate the effect of Reynolds number with dust particles to the turbulence. The dust particle distribution plays an important role in the solid-liquid two phase flow and it has a direct influence on the dynamic characteristics of the two-phase flow and the final sedimentation state of the dust particles. Discussing the effect of Reynolds number on dust particles, we first summarize the behavior of solid-liquid two phase flow on dust particle simulations, focusing particularly on dust particle-induced modifications to momentum fluxes across the domain. We performed a statistical average of the dust particle number corresponding to different positions in the wall-normal direction. The total volume of dust particles was then divided by the local mesh volume corresponding to different positions in the wall-normal direction. Fundamental understanding of the complex interactions between the solid-liquid dust particles and the turbulent carrier flow is of great significance for model development and design improvement of engineering devices. Therefore, dust particle –turbulence intersections, including the effect of Reynolds number of turbulent flow on dust particles (one-way coupling) and the effect of dust particles on fluid (two-way coupling) have attracted much attention from both physicists and engineers since 1960S.Experimental measurements, theoretical analysis and numerical modeling have been extensively applied to investigate solid-liquid dust particles-turbulence interactions and remarkable achievements have been made. For the effect of Reynolds number of turbulent flow on dust particles, one of the most striking features identified in turbulent flows laden with inertial dust particles has been the very strong in homogeneity of the dust particle concentration field. Experimental observation suggests that dust particles may augment or attenuate the turbulent kinetic energy of the carrier phase turbulence. Even the same dust particles were observed to have different effects of Reynolds number on turbulence at different regions of the flow field. The turbulence is attenuated because the motion of the dust particles drains energy from the mainstream. Dust particles used in these simulations are small so the wake behind dust particles is also very small and therefore has no significant effect. If the wake would be sufficiently great the turbulence would be generated by dust particles. Most of the dust particles accumulate along the circumference of eddies. That might be why the dust particles tend to move towards the centre of the vortexes under the action of an instantaneous drag force and the high pressure from the surrounding liquid. When these dust particles move inside the vortexes, they will rotate together with the vortexes. If the motions of the dust particles along with the vortexes occur under the ideal and perfect condition, these dust particles with definite mass, can be influenced by the centrifugal force caused by the rotary motion of the vortexes and escape from the vortexes again. Finally, these dust particles deposit in the appropriate places at which a balance occurs between the centrifugal force and the inter phase force. In existence, the mechanism responsible for the dust particle distribution is very complicated and depends on many factors such as the dust particle diameter, the turbulence structure, the flow direction against gravity, etc.

6. Conclusion

In this article, the effect of Reynolds number on turbulence modulations with presence of dust particles at solid-liquid two phase flow in a channel was investigated by Eulerian-Lagrangian model. Our Eulerian-Lagrangian simulations confirm that the separation process involves interplay between centrifugal forces induced by swirl and dispersion due to turbulence. In our calculations both swirl and turbulence are affected by the presence of dust particles, even at the relatively modest solid-liquid loadings that we have considered. . Most of the dust particles move closer to the centre of the channel; the local void fraction and the number of dust particles exhibit an approximately symmetric distribution along the central plane of the channel. Dust particles along the circumference of the vortex inhibit the vortex motion, which leads to turbulence modulation of the liquid. The investigations of the impact made by dust particles on the fine structure of turbulence of carrier flow in particular on the spectrum of turbulent fluctuations of velocity and on the micro scales of turbulence. The investigation results make it possible to predict the effect of dust particles on the turbulent energy of carrier flow. The control of the properties of continuum flows of flow trains of power plants by introducing dust particles of certain physical properties into the flow at certain concentration of dust particles may be very effective. However, one must bear in mind that the presence of dust particles in flows almost always entails the possibility of sedimentation of dust particles on the walls, erosion and other negative effects. In this article, we did not consider the influence of the shape of the dust particle size distribution on the turbulence performance; we only fed the turbulence with a uniform size distribution. I hope that this monograph will generate interest among students, post-graduates and researchers involved in investigations of hydrodynamics and heat transfer in solid-liquid two phase flow of dust particle-carrying turbulent flows and will give impetus to further development of the theory of multi phase flows.

References

[1] M.C.Ruzicka, J.Drahos, P.C.Mena, J, A. Teixeira. Effect of viscosity on homogeneous-heterogeneous flow regime transition in bubble columns. Chemical Engineering Journal96 (2003), 15-22.
[2] Ellis, L.B&Joubert, P.N.(1974):Turbulent shear flow in a duct.J.Fluid Mech.62,65-84.
[3] Wilson, P.L.(2003):On the Core flow and turbulent boundary layer in a curved duct. PhD thesis, University of London.
[4] Reynolds.O (1884):Phil.Trans.Roy.Soc. London, 123,186.
[5] Kishore, N. (1977):J.Scientific Research, BHU, 2,163.
[6] Sarker, M.S.A (1997): Journal of Energy Reserarch, 21, 1399.
[7] M.S.Alam Sarker and M.L.Rahman, Decay of turbulence before the final period in presence of dust particles, Rajshahi University Studies, Part-B, vol-25,121-132,(1997).
[8] Tandiroglu, A., Effect of Flow Geometry Parameters on Transient Heat Transfer for Turbulent Flow in a Circular Tube With Baffle Inserts. International Journal of Heat and Mass Transfer, 49(2006), 9-10, pp, 1559-1567.
[9] E.M.Sparrow, J.P.Abraham, W.J.Minkowycz., Flow Separation in a diverging conical duct: Effect of Reynolds number and divergence angle. International Journal of Heat and Mass Transfer, 52(2009), pp, 3079-3083.
[10] C.G.Speziale, R.M.C.SO, B.A.Younis., ON THE PREDICTION OF TURBULENT SECONDARY FLOWS, NASA Contractor Report 189722, ICASE Report No: 92-57.
[11] Kwok, K.C.S., Turbulence Effect on Flow around Circular Cylinder, Journal of Engineering Mechanics 112, 1181-1197, 1986.

Author Profile

Asaduzzaman received the B.Sc. (Honours) (First Class) in Applied Mathematics, Rajshahi University and M.Sc. (First Class) in Applied Mathematics, Rajshahi University in 2006 and 2007.