Journal of Zhejiang University SCIENCE
ISSN 1009-3095
http://www.zju.edu.cn/jzus
E-mail: jzus@zju.edu.cn

Influence of coherent structures in the gas-particle circular cylinder wake flow*

JI Feng (嵇 峰)†, LIU Lan (刘 兰), FAN Jian-ren (樊建人)†‡, CEN Ke-fa (岑可法)

(*Clean Energy & Environment Engineering Key Laboratory of Ministry of Education, Institute for Thermal Power Engineering, Zhejiang University, Hangzhou 310027, China*)

†E-mail: fengjiren@zju.edu.cn; fanjr@zju.edu.cn

Received Dec. 17, 2004; revision accepted Feb. 24, 2005

Abstract: To investigate the influence of coherent structures in the gas-particle wake flow, direct numerical simulation (DNS) method was adopted to compute a two-dimensional particle laden wake flow. A high accuracy spectral element method (SEM) was employed to simulate the gas flow field and a Lagrangian approach was used to compute the particles movement. Numerical results showed that at the same Stokes numbers, particles would be greatly impacted by the development of the coherent structure. But with different Stokes numbers, it can be seen that the large-scale vortex structures would influence the particle flow differently. While under different Reynolds numbers (150 and 200), there are no great changes in the particle laden flow.

Key words: Direct numerical simulation (DNS), Wake flow, Coherent structures, Particle laden flow
doi:10.1631/jzus.2005.A1132 **Document code:** A **CLC number:** TK16

INTRODUCTION

Turbulent gas-particle flows are frequently found in natural phenomena and industrial processes. Cases of cylinders in cross flows with particles occur in heat exchange equipment, including the convective zone of a fluidized-bed combustor, and in the primary superheaters, reheaters, and economizers of coal-fired boilers.

Coherent structures often occur in the above-mentioned gas-particle flow, and have great effect on such different systems, while some features of turbulent multiphase flow resemble those in single-phase flow. For example, in the typical occurrence of coherent structures, the stretching of the Karman vortex can be clearly seen in both single-phase flow and multiphase flow.

Many and various numerical models for multiphase turbulent flows were summarized by Crowe *et al*.(1996). In recent years direct numerical simulation (DNS) has become a powerful tool (Parviz and Krishnan, 1998) for the study of fluid dynamics, although it is sometimes used for solving some ideal and simple problems far from technical application level.

Recent reports to predict the three-dimensional features of particle dispersion for a plane mixing layer were made by Fan (2001; 2003) and Marcu and Meiburg (1996). Simulation results showed that streamwise vortices produce additional effects that modify the dispersion patterns of particles. Along the spanwise direction, particle dispersion also develops into ‘mushroom’ patterns, owing to intense 3D vortex stretching and folding.

Chung and Troutt (1988) simulated the particle dispersion in an axisymmetric jet using a discrete vortex element approach with the interesting outcome was that the ratio of particle dispersion to fluid agreed well with experimental results.

According to the geometric complexity

---

‡Corresponding author
* Project (No. 50236030) supported by the National Natural Science Foundation of China

(Karniadakis, 1999), wake is more complex to simulate than other shear flows (mixing layer and jet). Slater and Young (2001) studied the particle flow over a circular cylinder by using an Eulerian formulation. This paper presents the seldom studied before the coherent structures effect on complex gas-particle flow. DNS for calculating particle-laden wake flow was done in this paper, with emphasis placed on clarifying the impact mechanism of coherent structures in the wake flow.

MATHEMATICAL MODEL

Flow configuration and simulation

In the direct numerical simulation, we adopted spectral element method to solve the gas flow field. The computational domain was set to 14×37 (Fig.1). And the computational time step was 0.01.

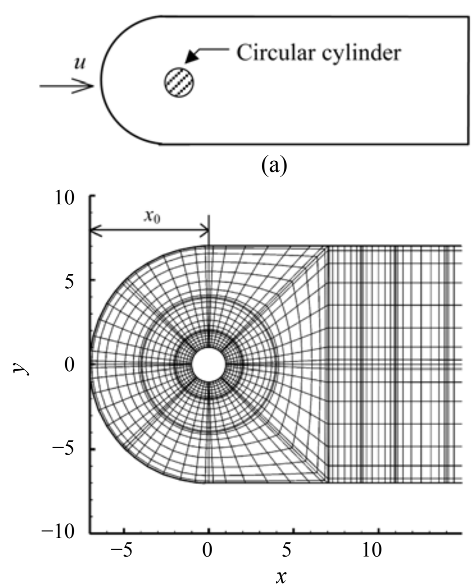


Fig.1 (a) Sketch of computational domain; (b) Partial view of computational mesh

Incompressible viscous fluid flow N-S equations and conservative forms are normalized as follows:

$$\frac{\partial \boldsymbol{V}}{\partial t} = -\nabla p + \frac{1}{Re}\boldsymbol{L}(\boldsymbol{V}) + \boldsymbol{N}(\boldsymbol{V}) \quad \text{in}\ \Omega \qquad (1)$$

$$\nabla \cdot \boldsymbol{V} = 0 \quad \text{in}\ \Omega \qquad (2)$$

where $\boldsymbol{V}$ is the velocity, p is pressure (density ρ=1). The Reynolds number Re is defined as $Re=U_0 d/\nu$, with U_0 the uniform stream velocity, d characteristic length, and ν kinematic viscosity. Linear dispersion operator and nonlinear convection operator are:

$$\boldsymbol{L}(\boldsymbol{V}) = \nabla^2 \boldsymbol{V}, \quad \boldsymbol{N}(\boldsymbol{V}) = -[\boldsymbol{V}\cdot\nabla\boldsymbol{V} + \nabla\cdot(\boldsymbol{V}\boldsymbol{V})]/2$$

respectively represented in oblique symmetric form to reduce the anti-aliased error.

A detailed description of the computational approach and its validation is given in (Yao *et al*., 2003). In this paper we focus on the particle movement analysis.

Particle movement computation

Assumptions: First, all particles are rigid spheres with the same diameter and density. Second, at the initial time, particles are distributed uniformly in the flow and have the same dynamic characteristics. Since the density ratio ρ_p/ρ_g>1000, in the dilute two-phase flow the Basset force and the added force can be neglected. The influence of the particles on the gas-flow and particle-particle interactions is neglected because of the low particle loading.

In addition, the Saffman force and Magus force are neglected in this simulation too. So, only the most important drag force due to the relative velocity between the phases is taken into account in this study of the particle dispersion.

Therefore, the non-dimensional motion equation based on a particle is expressed as:

$$\mathrm{d}\boldsymbol{V}/\mathrm{d}t = f/St(\boldsymbol{U}-\boldsymbol{V}) \qquad (3)$$

where $\boldsymbol{V}$ is the particle velocity, $\boldsymbol{U}$ is the fluid velocity at that particle position and f is a modification factor for the Stokes drag coefficient, which is described by $f = 1 + 0.15Re_p^{0.678}$, with $Re_p=|\boldsymbol{U}-\boldsymbol{V}|d_p/\nu$. St named Stokes number for a particle is defined as: $St=(\rho_p d_p^2/18\mu)/(R/U_0)$. The velocity and position of particles can be obtained by integrating Eq.(3).

Particle-wall collision model

Grant and Tabakoff (1975) obtained the expression of particle-wall collision by practical experiments. The change in particle momentum due to impact was found to be mainly a function of the particle impact velocity and its incidence angle.

$$V_{n_2}/V_{n_1}=1.0-0.4159\beta_1-0.4994\beta_1^2+0.292\beta_1^3 \quad (4)$$

$$V_{t_2}/V_{t_1}=1.0-2.12\beta_1+3.0775\beta_1^2-1.1\beta_1^3 \quad (5)$$

where V_n and V_t represent the particle velocity impact components normal and tangential to the wall, respectively. Subscripts 1 and 2 refer to the conditions before and after impact, respectively. In the above equations, β_1 (in radians) is the angle between the incident velocity and the tangent to the surface which is supposed to be smooth.

So, if the detailed velocity history was known for particle that ultimately strikes a surface, we can compute the impact velocity from which the corresponding incidence angle and incidence speed can be computed.

The Lagrangian approach yields a more detailed physical description of the particle phase (such as individual particle speeds, trajectories, and residence times), compared to Eulerian description. The particle trajectories are determined using an adaptive step size fourth order Runge-Kutta method.

NUMERICAL RESULTS AND DISCUSSION

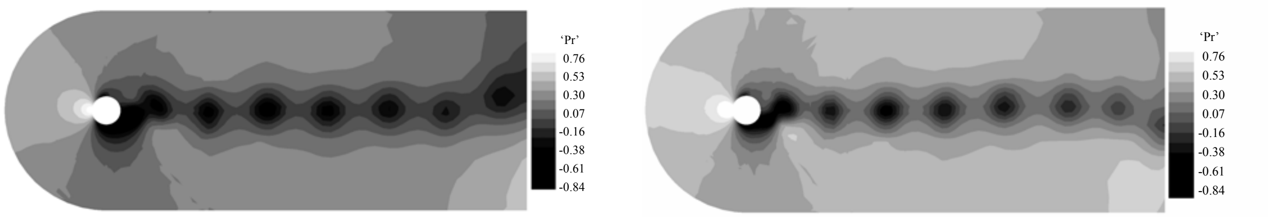
Fig.2 of the pressure contour for different Reynolds numbers in the whole computational domain shows clearly the flow characteristics of circular cylinder wake.

The recirculating separation region shows very low pressure in both instances. Obviously, most pressure values of *Re*=200 are higher than those of *Re*=150 according to the iso-pressure contour view. Lowest pressure exists in the recirculating regime because of the highest velocity existing there.

The analysis of the results has to be considered with respect to the operative *St* number.

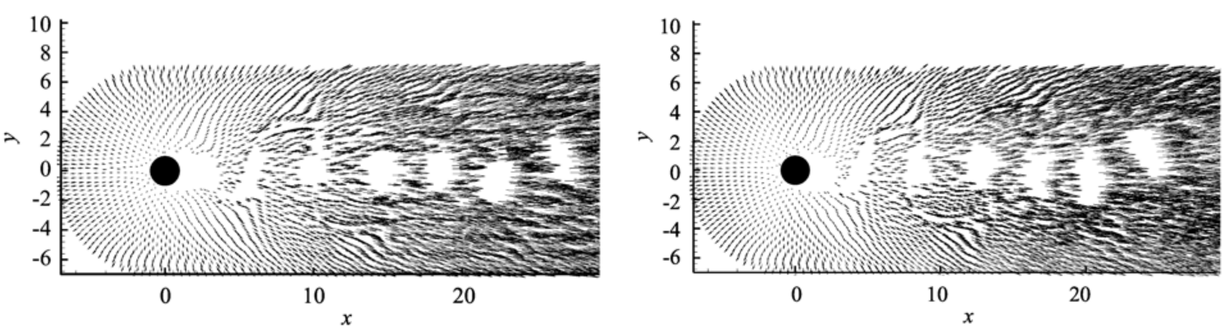
In Fig.3 and Fig.4, the results indicate that coherent structure exhibits in the particle flow field as well as in the gas flow field.

Particles with small Stokes number (*St*=0.1) have low inertial and very rapid response to changes in fluid velocity. The particle vector field shows that some particles had entered the vortex core. In contrast, particles with medium Stokes number (*St*=1) are faithfully follow the flow and, in principle, can be used to visualize fluid motion.



(a) (b)

Fig.2 Comparative view of the pressure field for Reynolds numbers (instantaneous snapshot, *T*=100)
(a) *Re*=150; (b) *Re*=200



(a) (b)

Fig.3 Comparative view of the particle vector field for Reynolds numbers (instantaneous snapshot, *T*=100)
(a) *Re*=150; (b) *Re*=200, *St*=0.1

Instantaneous particles movement (Fig.3 and Fig.4) shows that the small particles enter the inner regions of the vortices but that the bigger cannot do the same. The following vertical dispersion function is introduced (Ling *et al*., 1998).

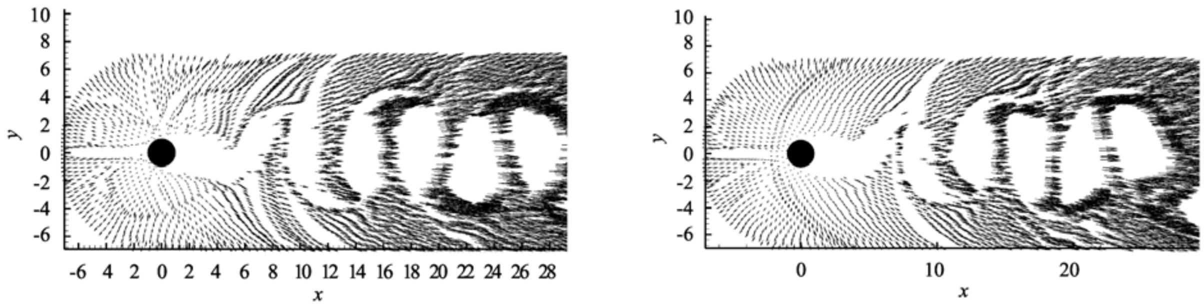
$$D_{\mathrm{y}}(t)=\left(\sum_{i=1}^{n_{\mathrm{t}}}(Y_i(t)-Y_{\mathrm{m}}(t))^2/n_{\mathrm{t}}\right)^{1/2}$$

where n_t is total particle number in the computational domain when time is t, $Y_i(t)$ is the ith particle instantaneous vertical displacement, $Y_{\mathrm{m}}(t)$ is mean vertical displacement of total particles at that time. This function reflects the dispersion degree.

Particle dispersion function in vertical direction is quantitatively and instantaneously plotted in Fig.5 showing that decreases with increase of particle Stokes number. For the circular cylinder wake, two symmetrical vortices with opposite sign are separately and alternately generated. They are convected and diffused away from the cylinder but no vortex pairing interactions occur. The mechanism for the particle dispersion in circular cylinder wake mainly depends on the repulsion force associated with the vortex sheet regions between two adjacent vortex structures with opposite sign, which is named stretching process.

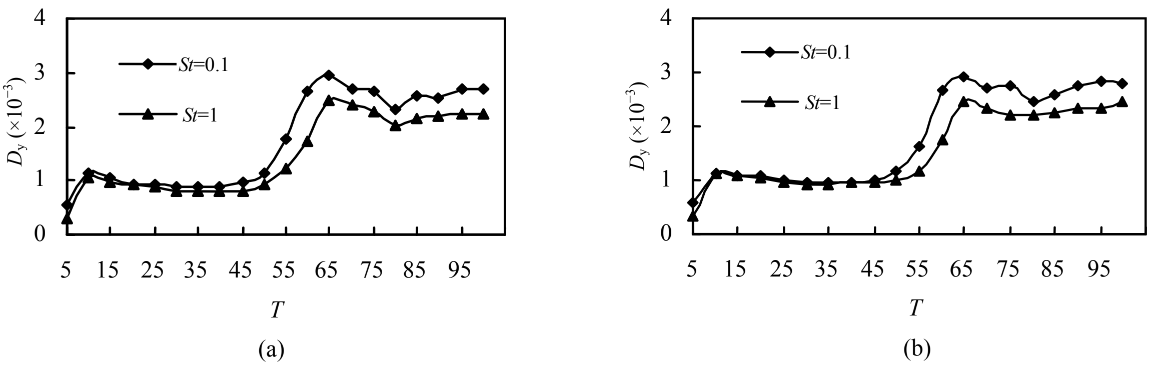
That is to say, it is obvious that particles follow the gas flow and do not collide with the front face of the cylinder when the particle size is small. But when *St* equals 1, almost no particle enters the core. These results showed that particle flow of medium Stokes number is more influenced by gas flow than the small Stokes number.

When *Re* equals 200, the perturbation is stronger than that when *Re* equals 150. So particles flow dynamic characteristics have different aspects. But under the different Reynolds numbers 150 and 200, there are no great changes in the particles flow. For examples, when both of Stokes numbers are 0.1, the irregularities of particle flow vortex structure are similar.



(a) (b)

Fig.4 Comparative view of the particle vector field for Reynolds numbers (instantaneous snapshot, *T*=100)
(a) *Re*=150; (b) *Re*=200, *St*=1



(a) (b)

Fig.5 Time-dependent particle dispersion in the vertical direction
(a) *Re*=150; (b) *Re*=200

CONCLUSION

We have presented a numerical study of gas particle wake flow. Simulations were performed to gain more insight into the flow dynamics than could be obtained from previous experiments. For the numerical integration of the governing equations a high-order spectral element method (SEM) was employed.

The present results show that the change trends of particle flow are sometimes similar with different Reynolds numbers. Stokes numbers have determining influence on overall features of the gas flow.

Future work needs to be done in several areas. There is more to be done in studying the feedback of particles on gas flow, after which, we can investigate how particles influence the coherent structures of gas flow (called two-way coupling).

References

Chung, J.N., Troutt, T.R., 1988. Simulation of particle dispersion in an axisymmetic jet. *J. Fluid Mech.*, **186**:199-222.

Crowe, C.T., Troutt, T.R., Chung, J.N., 1996. Numerical models for two-phase turbulent flows. *Annu. Rev. Fluid Mech.*, **28**:11-43.

Fan, J.R., Zheng, Y.Q., Yao, J., Cen, K.F., 2001. Direct simulation of particle dispersion in a three-dimensional temporal mixing layer. *Proc. R. Soc. Lond. A*, **457**: 2151-2166.

Fan, J.R., Luo, K., Zheng, Y.Q., Jin, H.H., Cen, K.F., 2003. Modulation on coherent vortex structures by dispersed solid particles in a three-dimensional mixing layer. *Physical Review E.*, **68**:036309.

Grant, G., Tabakoff, W., 1975. Erosion prediction in turbomachinery resulting from environmental solid particles. *J. of Aircraft*, **12**:471-478.

Karniadakis, G.E., 1999. Simulating turbulence in complex geometries. *Fluid Dynamics Research*, **24**:343-362.

Ling, W., Chung, J.N., Troutt, T.R., Crowe, C.T., 1998. Direct numerical simulation of a three-dimensional temporal mixing layer with particle dispersion. *Journal of Fluid Mechanics*, **358**:61-85.

Marcu, B., Meiburg, E., 1996. Three-dimensional features of particle dispersion in a plane mixing layer. *Phys. Fluids*, **8**:2266-2268.

Parviz, M., Krishnan, M., 1998. Direct numerical simulation: A tool in turbulence research. *Annu. Rev. Fluid Mech.*, **30**:539-578.

Slater, S.A., Young, J.B., 2001. The calculation of inertial particle transport in dilute gas-particle flows. *International Journal of Multiphase Flow*, **27**: 61-87.

Yao, J., Feng, J., Liu, L., Fan, J.R., Cen, K.F., 2003. Direct numerical simulation on the particle flow in the wake of circular cylinder. *Progress in Natural Science*, **13**(5): 379-394.