Scientific Visualization, 2025, volume 17, number 2, pages 36 - 43, DOI: 10.26583/sv.17.2.03

Numerical Visualization of Vortex Wakes Behind Large Particles

A.A. Mochalov[1,A,B], A.Yu. Varaksin[2,A,B]

[A] Bauman Moscow State Technical University, Moscow, Russia
[B] Joint Institute for High Temperature, Moscow, Russia

[1] ORCID: 0000-0003-3078-1277, artem.mochalov@yandex.ru
[2] ORCID: 0000-0002-8799-6378, varaksin_a@mail.ru

Abstract

An attempt has been made to visualize the flow formed in the wake of large particles moving in an ascending turbulent air flow in the channel. Numerical modeling was performed using a simplified version of the approach called "two–way coupling" (TWC) in English literature and taking into account the inverse effect of particles on gas characteristics. The particle motion was calculated in an approximate manner, therefore the method used is called "quasi – two–way coupling", TWC(Q). The results of numerical modeling of the characteristics of turbulent trails behind large moving particles based on the Reynolds averaged Navier-Stokes equations (RANS) are presented.

Keywords: visualization, numerical simulation, two – phase flows, turbulent wake, turbulence, RANS.

1. Introduction

The features of the motion of a dispersed impurity in the form of particles in turbulent gas flows and its inverse effect on the turbulence characteristics of the carrier phase are key problems of the theory of two-phase flows [1-4]. The inverse problem is to study the effect of particles on the characteristics of the gas stream carrying them. The solution to this problem involves determining the characteristics of a gas in the presence of particles: velocity and temperature fields, friction and heat transfer coefficients, etc. [5-7]. The flow with large particles is characterized by the fact that the relaxation time of particles significantly exceeds the characteristic time of large-scale turbulent vortices, i.e. $Stk_L \to \infty$. Such particles will not react to the turbulent pulsations of the velocity of the carrier phase, and the distributions of their averaged velocities will be almost uniform along the section of the channel (pipe). The data [8] can serve as a clear confirmation of this.

The purpose of this work is to visualize the flow formed in the wake of large particles moving in an ascending turbulent air flow in a channel based on numerical modeling.

2. Modeling of the carrier phase

A characteristic feature of turbulent flows is the presence of random fluctuations in all flow parameters. Due to the variability of parameters not only in space, but also in time, various methods of averaging and smoothing are used in the study of turbulent flows, allowing to move from probabilistic fields of characteristics to their regular average values.

A complete system of Reynolds averaged equations (RANS) describing the motion of a viscous fluid in tensor form:

$$\begin{cases} \frac{\partial \rho}{\partial t} + \frac{\partial \left(\rho u_j \right)}{\partial x_j} = 0 \\ \frac{\partial \left(\rho u_i \right)}{\partial t} + \frac{\partial \left(\rho u_i u_j \right)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\tau_{ij} + \tau_{\text{т},ij} \right) \\ \frac{\partial \left(\rho E \right)}{\partial t} + \frac{\partial \left(\rho u_j H \right)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[u_i \left(\tau_{ij} + \tau_{\text{т},ij} \right) - \left(q_j + q_{\text{т},j} \right) \right] \\ \rho = \frac{p}{RT} \end{cases}, \quad (1)$$

where x_j – Cartesian coordinates; u_i, u_j – components of the velocity vector of the averaged flow, E – specific total energy, $H = c_p T + u_j^2/2$ - specific total enthalpy of a gas, T – temperature, ρ - gas density, p – pressure, R – gas constant.

Components of the Reynolds stress tensor $\tau_{\text{т},ij} = -\rho \overline{u_i' u_j'}$ and the Reynolds heat flux vector $q_{\text{т},i} = -\rho c_p \overline{u_i' T'}$ They appear when averaging nonlinear convective terms of the initial Navier-Stokes equations and energy transfer, and their relationship with the parameters of the averaged flow is unknown.

Since the RANS equations are not closed due to the Reynolds stress tensor and the turbulent heat flow vector, it is necessary to use additional relations (turbulence models) linking these values with the characteristics of the averaged flow.

In this paper, the SST k-ω turbulence model, the Menter model, is used to close the Navier-Stokes equations averaged by Reynolds [9]. This turbulence model was chosen because there are no additional (source) terms in the transfer equations of the two-parameter Menter model, which take into account the generation of turbulence by large particles.Actually, the additional generation of turbulence energy does not occur due to an additional source term in the equation, but due to the fact that the flow around large particles is calculated directly as objects with solid walls.

The Mentor model is implemented in the following form:

$$\begin{aligned} &\frac{\partial \left(\rho k \right)}{\partial t} + \frac{\partial \left(\rho u_j k \right)}{\partial x_j} = \frac{\partial}{\partial x_i} \left[\Gamma_k \frac{\partial k}{\partial x_i} \right] + G_k - Y_k, \\ &\frac{\partial \left(\rho \omega \right)}{\partial t} + \frac{\partial \left(\rho u_j \omega \right)}{\partial x_j} = \frac{\partial}{\partial x_i} \left[\Gamma_\omega \frac{\partial \omega}{\partial x_i} \right] + G_\omega - Y_\omega + D_\omega, \end{aligned} \quad (2)$$

where G_k - generation of turbulence kinetic energy k, G_ω – generation of turbulence kinetic energy dissipation ω per unit k, Y_k – dissipation of turbulence kinetic energy, Y_ω – dissipation ω, $\Gamma_k = \mu + \frac{\mu_\text{т}}{\sigma_k}, \Gamma_\omega = \mu + \frac{\mu_\text{т}}{\sigma_\omega}$, D_ω - the cross-diffusion term.

To solve the problem, a computational grid with 1.11 million cells was built. The study of grid convergence was carried out in terms of the sufficiency of the thickening of the grid in particles. The value of y+ in the flow of particles does not exceed 3, which suggests that the first cell is located in a viscous sublayer, wall functions are not used.

3. Modeling of large particles

The authors of the work chose the Ansys Fluent software to visualize the flow in the wake of large particles, since this software is a powerful tool for modeling complex problems in the field of hydrogas dynamics. Ansys Fluent allows you to model the behavior of liquids and gases in various conditions, including laminar and turbulent flows, taking into account viscosity, density and other properties, and also has many tools for visualizing calculation results [10].

Modeling of large particles is implemented using the module Dynamic Mesh in Ansys Fluent. Smoothing was carried out using the method Spring/Laplace/Boundary Layer – this is a smoothing function when rebuilding the calculated grid. In the method used, the edges of the cells are represented as springs and a coefficient is set, which is actually the proportionality coefficient in Hooke's law, thereby it is possible to regulate the thickening of the grid during rebuilding. The dynamic grid tracks the movement of particles and is rebuilt, smoothing functions are used to ensure that the quality of the grid remains close to the initial one. The grid was rebuilt using the local cell method with a minimum scale of 0.0005 m and a maximum scale of 0.1 m. The movement of particles was set by one degree of freedom using the Six DOF (degree of freedom) module, which allows calculating forces and moments acting on an object that lead to a change in its position in space with a given time step. The paper assumes that particles have one degree of freedom – they can only move along the Y axis without rotation. Accordingly, in addition to the boundary and initial conditions for the particles, a mass is set to take into account gravity (in the work, the mass of each particle is 0.5 g).

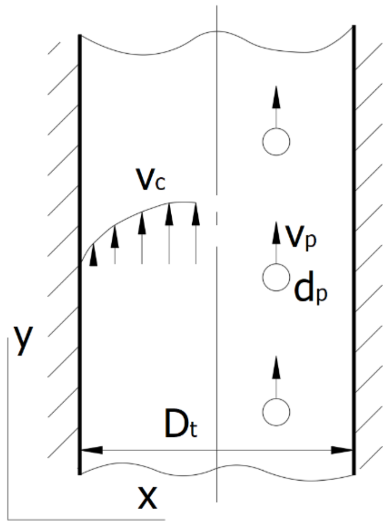

Fig.1 The scheme of particle motion in the carrier gas

Figure 1 shows a diagram of the movement of particles at a speed of v_p in the carrier gas v_c. The center of the rectangular coordinate system (x – y) it is located on the axis of symmetry of the channel with a diameter of D_t. Large particles are spheres with a diameter of d_p, which are placed vertically one after the other along the length of the channel L at different distances from each other in order to exclude the mutual influence of vortex traces behind the spheres. The spheres are located with an offset relative to the axis of symmetry of the channel by 5 mm.

Table 1. Main characteristics of the studied flow

Air velocity v_c, m/s	Particle velocity v_p, m/s	Particle diameter d_p, m	Pipe diameter D_t, m	Pipe length L, m	Reynolds number Re_p	Reynolds number Re_c
14.9	5.7	0.003	0.0305	0.2	1120	$3 \cdot 10^4$

4. Visualization of vortex traces behind large particles by TWC(Q) method

For a detailed analysis of the process of additional generation of air turbulence energy in a sample section of the pipe, an original methodological technique was proposed, consisting in the arrangement of particles on the same line (Fig. 1).

Using this technique, it was possible to obtain visual distributions of turbulence energy along the length of the channel and identify qualitatively different areas (Fig. 2): 1) the initial section characterized by increased values of turbulence energy of "clean" air (single-phase flow); 2) areas of growth of turbulence energy of the gas phase behind individual particles; 3) the area of "quasi-stationary" single-phase flow.

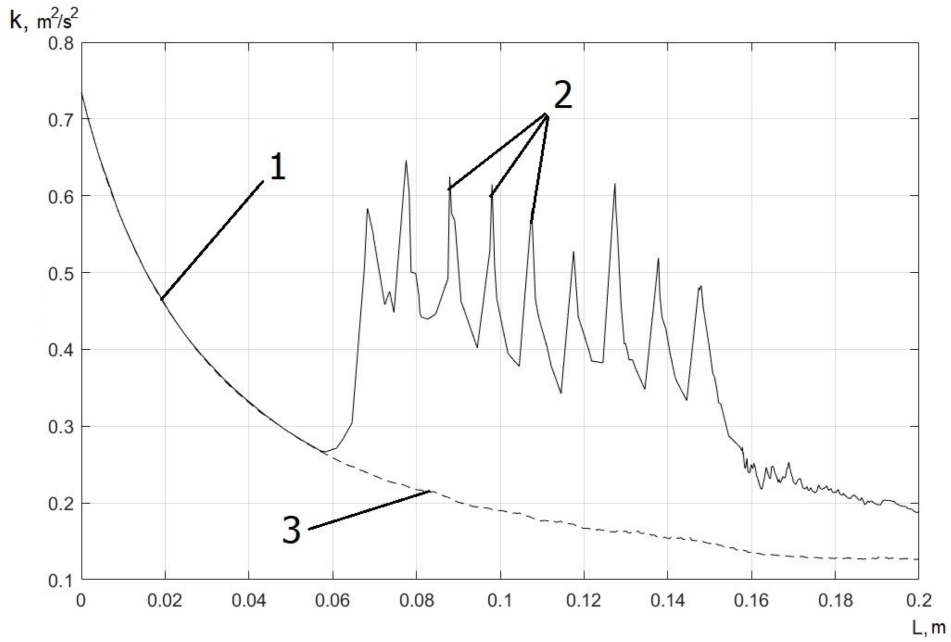

Fig. 2 Distribution of turbulence energy k according to the length of the channel for $N_p = 10$, The numbers are indicated by: 1 – the initial section with an increased k; 2 – growth k in the case of individual particles; 3 – "quasi-stationary" single-phase flow

The data in Fig. 2 indicate that developed turbulent traces are formed behind large particles, which are characterized by the presence of non-stationary three-dimensional vortex structures.

Figures 3-5 show typical turbulence energy distributions k, in the carrier gas in the channel and in the traces of the particles. The visualization of the gas and particle flow is performed in the Ansys Fluent software. It should be noted that the equations of motion of the particles were not integrated, and the particles themselves moved at the same speeds. Thus, a simplified version of the approach, called "two–way coupling" in English literature, was implemented, i.e. taking into account the inverse effect of particles on gas characteristics. Let's call it a quasi–approach or, using the English abbreviation, "quasi – two-way coupling", TWC(Q). In this setting, when the calculation of the flow of gas around each single particle is performed and the interphase boundary is resolved (such calculations are called "particle – resolved" (PR)). Thus, the approach used in this work can be classified as PR – TWC(Q) – RANS.

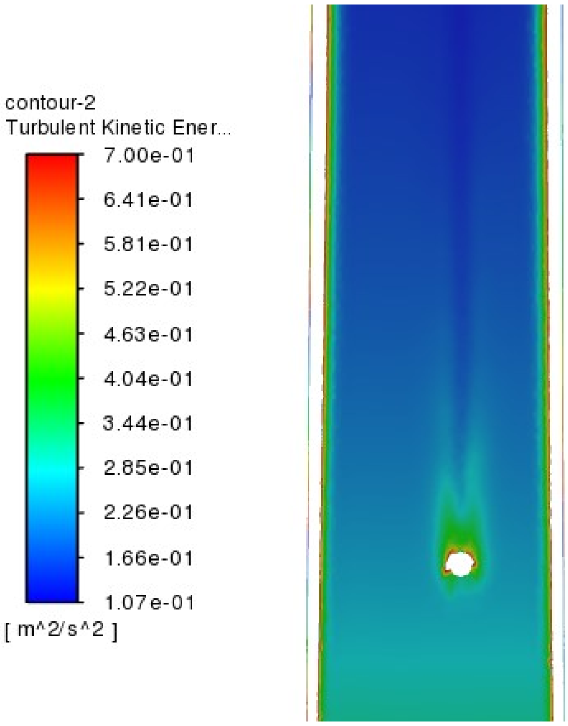

Fig. 3 Distribution of the turbulence kinetic energy behind a single particle, N_p = 0.1, 1/ cm³

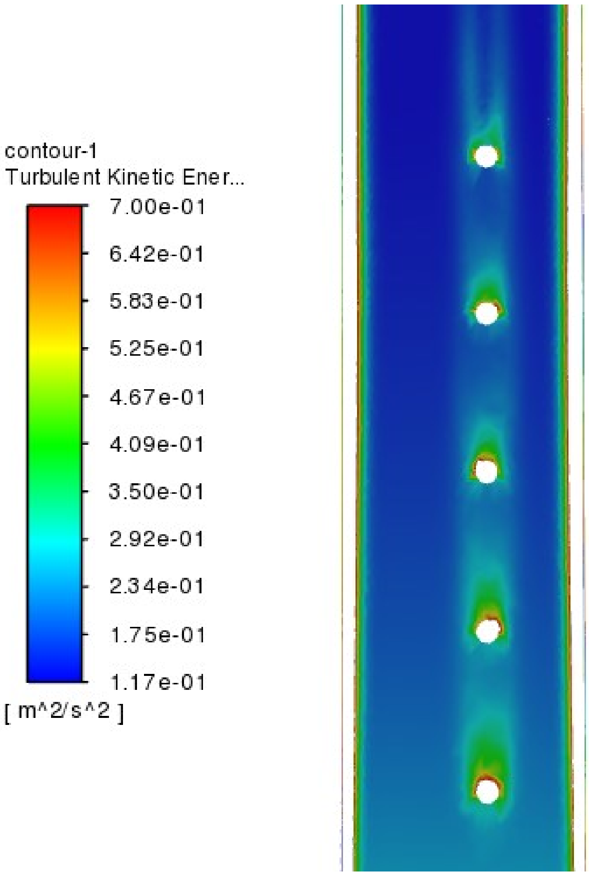

Fig. 4 Distribution of the turbulence kinetic energy behind a group of particles, N_p = 0.5, 1/cm³

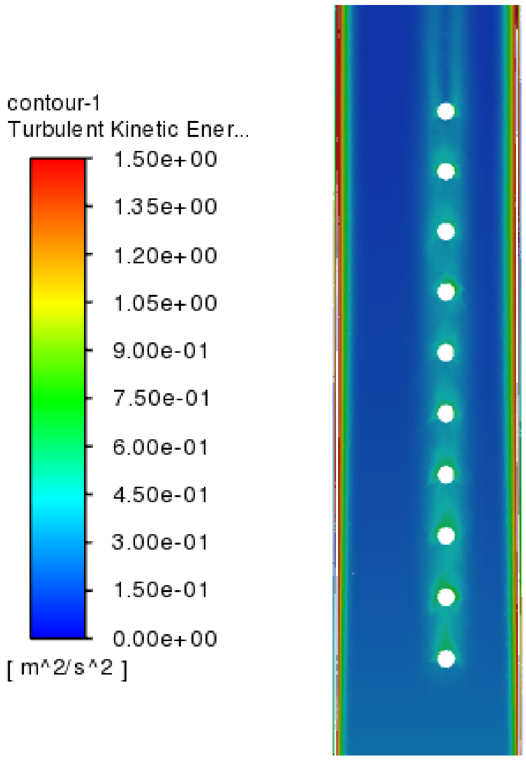

Fig. 5 Distribution of the turbulence kinetic energy behind a group of particles, N_p = 1, 1/cm^3

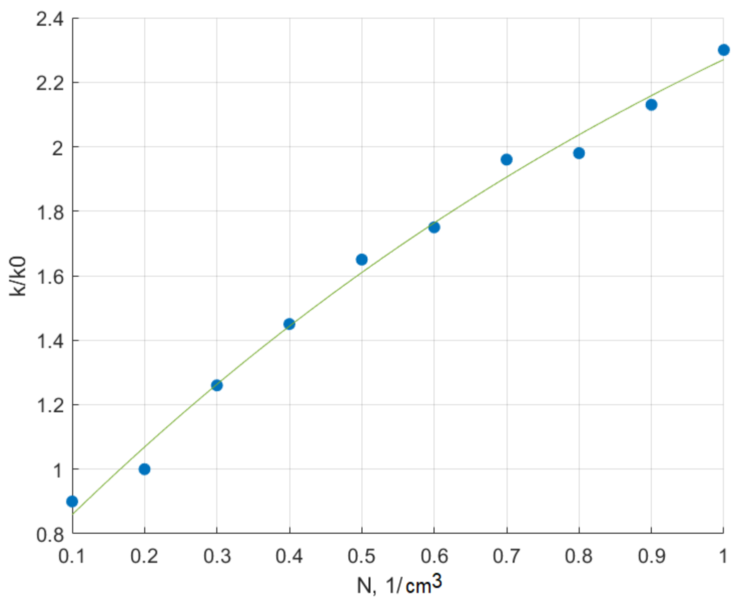

Fig. 6 Growth in turbulence energy generation depending on the calculated particle concentration

Figure 6 shows data on the increase in turbulence energy of the carrier gas from the value of the calculated particle concentration (the number of particles per unit volume).

5. Conclusion

The paper visualizes the flow behind large particles moving in an ascending turbulent air flow in the channel. Numerical simulation is performed using a simplified version of TVC(Q). Some results of numerical modeling of the characteristics of turbulent traces behind large moving particles based on Reynolds averaged Navier-Stokes equations (RANS) are presented.

The dependence of the turbulence energy value on the calculated particle concentration is revealed.

Acknowledgements

The work is supported by the Russian Scientific Foundation (Project № 23-19-00734).

References

1. Crowe C., Sommerfeld M., Tsuji Y. (Eds.). Multiphase Flows with Droplets and Particles. Boca Raton, FL, USA: CRC Press, 1998. 471 p.
2. Michaelides E.E., Crowe C.T., Schwarzkopf J.D. (Eds.). Multiphase Flows Handbook, 2nd ed. Boca Raton, FL, USA: CRC Press, 2017. 1396 p.
3. Zaichik L.I., Alipchenkov V.M., Sinaisky E.G. Particles in Turbulent Flows. Darmstadt, Germany: Wiley-VCH, 2008. 320 p.
4. Varaksin A.Y. Collisions in Particle-Laden Gas Flows. New York, NY, USA: Begell House, 2013. 370 p.
5. Varaksin A.Yu. Hydrogasdynamics and Thermal Physics of Two-Phase Flows with Solid Particles, Droplets, and Bubbles. High Temperature, 2023, vol. 61, p. 852–870.
6. Yu Z.S., Xia Y., Guo Y., Lin J.Z. Modulation of Turbulence Intensity by Heavy Finite-Size Particles in Upward Channel Flow. J. Fluid Mech., 2021, vol. 913, paper no. A3.
7. Yang B., Peng C., Wang G.C., Wang L.P. A Direct Numerical Simulation Study of Flow Modulation and Turbulent Sedimentation in Particle-Laden Downward Channel Flows. Phys. Fluids, 2021, vol. 33, paper no. 093306.
8. Varaksin A.Yu., Mochalov A.A., Zhelebovsky A.A. Flow Characteristics in the Wake of a Large Moving Particle. High Temperature, 2022, vol. 60, p. 639–644.
9. Menter F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA J., 1994, vol. 32, p. 1598–1605.
10. Басов К.А. Ansys для конструкторов. – М.: ДМК Пресс, 2009 – С. 248