29th Congress of the International Council of the Aeronautical Sciences
St. Petersburg, Russia; September 7–12, 2014

COMPUTATIONAL SIMULATION OF TWO-PHASE GAS-PARTICLE FLOW IN AN INLET "ROTOR-STATOR" STAGE OF A TURBOJET ENGINE COMPRESSOR

Yu.M. Tsirkunov, D.A. Romanyuk, S.V. Panfilov
Department of Plasma- and Gas Dynamics and Heat Engineering
Baltic State Technical University,
1st Krasnoarmeiskaya str. 1, Saint Petersburg 190005, Russia

Keywords: *gas-particle flow, airfoil cascade, particle size distribution, particles scattering, inter-particle collisions*

Abstract

Two-phase gas-particle flow through two, moving and immovable, cascades of airfoils (blades) is studied numerically. The carrier gas is treated as a continuum. Its flow is considered as two-dimentional and described by the Navier–Stokes equations (pseudo-DNS approach) and the Reynolds averaged Navier–Stokes equations (unsteady RANS approach) with the Menter $k-\omega$ SST turbulence model. The governing equations in both cases are solved by CFD-methods. The dispersed phase is treated as a discrete set of a large number of solid particles. The Lagrangian method for collisionless particle phase or the direct simulation Monte Carlo (DSMC) method for flow with inter-particle collisions is used in computational simulation. The effects of gas-particle interaction, particle size distribution, particle scattering in particle-blade collisions, and inter-particle collisions are investigated separately and in combination.

1 Introduction

Aircraft engines are not designed for operating in a dusty environment but military aircrafts can often take off, fly and land in an atmosphere containing dust or sand. Civilian aircrafts can also travel through a cloud of volcanic ash. In all these cases, engines ingest suspended solid particles, which exert a negative effect on engine performance. Dispersed particles in a high-speed inlet flow, being more inertial than the carrier gas, collide with blades of fan, compressor and turbine resulting in their abrasive erosion. Particles may also deposit on hot surfaces of turbine components and nozzle, blockage cooling passages etc. These and some other phenomena are described and discussed in recent reviews [1, 2].

The problem of quantitative prediction of particle effect on engine operation and its lifetime is multi-physical and very complex. It has been the focus of attention of researchers for the last 40 years. Modelling and simulation of dusty gas flow through compressor and turbine rows of blades are the most important parts of the whole problem. Strictly speaking, such a flow is time-dependent and three-dimensional, however many key flow features in the sequential rows can be studied using a 2D flow model in the plane representing a developed mean-circle cross section. This approach is well known as the 2D theory of cascade flow.

The main aim of the present study was to clarify the role of different random by nature phenomena in redistribution of particles in the flow. The following effects were taken into account: the particle size distribution, the particle scattering in particle blade collisions due to non-spherical particle shape, and collisions between the particles. We consider 2D flow model for the carrier gas to be preferable for this purpose, because taking into account three-dimensional effects has a negligible influence on particles motion [3], and makes relevance analysis unduly difficult.

Although the 2D carrier gas flow model was used, the motion of particles was considered in 3D formulation.

In the present paper, the carrier gas and particle-phase flow models are described very briefly with references to the original papers and particular attention is paid to computational results and discussion.

2 Arrangement of Cascades, Flow Scheme and Parameters

We consider a two-phase gas-particle flow through a set of two cascades of airfoils (blades) as shown in Fig. 1. The first cascade (rotor) moves at constant velocity V_r in the direction normal to an undisturbed flow, and the second cascade (stator) is fixed. Both cascades have the same pitch s. The rotor blades are set at the angle β. In the computational simulation, a particle cloud of finite width h is considered to visualize the particle-phase flow.

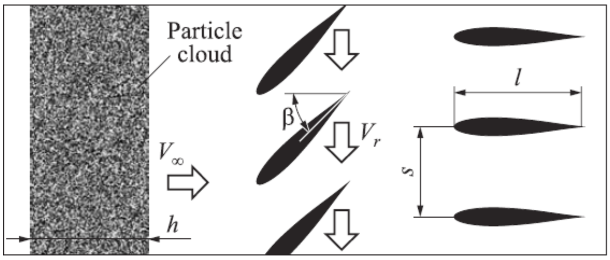

Fig. 1. Arrangement of cascades and schematic of flow.

We consider a two-phase flow as one-way coupled, taking account of gas-particle interaction in simulation of particles motion, and ignoring the reverse effect of the dispersed phase on the carrier gas flow. In this case, the complex gas-particle flow problem is splitted into two sequential steps: modelling of the carrier gas flow, and calculation of the particles motion in the predetermined gas flow field. Although the gas flow is two-dimensional, the motion of every particle is considered as three-dimensional because 3D effects are important for the inter-particle collisions and scattering of particles in particle-blade collisions.

Parameters of the cascades and flow properties in calculations were taken close to typical for an inlet compressor of a turbojet engine. The airfoil chord l was 10 cm, the cascade pitch s was 7 cm, and the angle β was 45°. The NACA0012 profile was used for both rotor and stator blades. In gas flow calculations, the inlet velocity ranged from the value corresponding to zero velocity in undisturbed flow to the value corresponding to cruise flight ($V_\infty = 200$ m/s). The rotor cascade velocity V_r was 150 m/s. The carrier gas was air. The particle material was silicon dioxide.

3 Modelling of the Carrier Gas Flow

The flow through cascades is usually turbulent. In the present study, two models were used for simulation of the carrier gas flow: the complete Navier–Stokes equations (that corresponds to the pseudo-DNS approach because the flow was considered as 2D, and the grid was too coarse for high resolution of fine turbulent flow structure), and the unsteady Reynolds averaged Navier–Stokes equations (URANS approach) with the Menter $k-\omega$ SST turbulence model. These carrier gas flow models and CFD-simulation technique are described in detail in papers [3, 4, 5] and not given here.

It should be noted that the URANS approach results in averaged in time gas flow field. However, in multiphase flow not averaged but actual gas flow parameters determine the interphase interaction and, hence, the disperse phase behaviour. Simulation of the carrier gas flow on the basis of the Navier–Stokes equations gives more detailed flow structure than the RANS equations (even on a coarse grid), which is important for calculation of particles motion. The carrier gas flow structure is visualized in Fig. 2 by the entropy function fields ($S = p/\rho^\gamma$) and by the patterns of massless markers. It is seen that rather complex vortex structure of the flow (left) is blurred when simulated using the URANS approach (right).

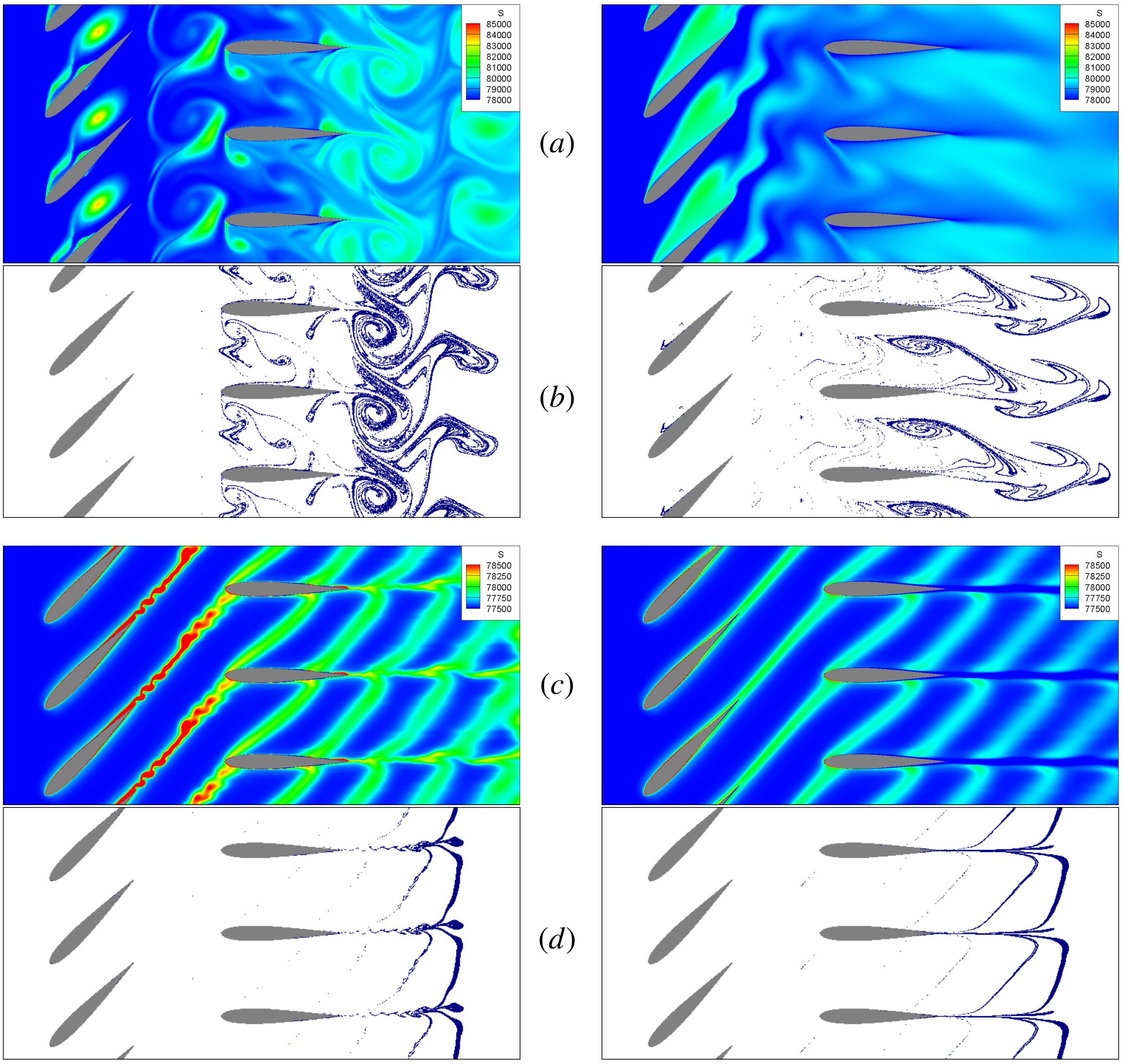

Fig. 2. Instantaneous fields of entropy function $S = p/\rho^{\gamma}$ (a, c) and corresponding patterns of marker cloud (b, d) obtained using the Navier–Stokes flow model (left) and the $k-\omega$ SST turbulence model (right) for the flows with inlet velocity of 100 m/s (a, b) and 200 m/s (c, d).

4 Modelling of the Particle-Phase Flow

The motion of particles is governed by the gas-particle interaction, particle-blade collisions and inter-particle collisions.

The gas-particle interaction model includes the drag force, the lift Magnus force and the torque. These factors dominate over all other force components in the flow under consideration. The last two are significant for rapidly rotating particles, for example, after particle-blade collision. A detailed description of the gas-particle interaction model is given in [3, 4, 5].

Dynamic sensitivity of a particle to the fine structure of the carrier gas flow depends on its relative inertia described by the Stokes number $\mathrm{Stk} = \rho_p d_p^2 V_\infty/(18\mu_\infty l)$ which is defined as the ratio of the particle dynamic relaxation length (using the Stokes law for a particle drag force) to the flow characteristic length l. In calculations, we ranged particle size from 10 to 40 μm, thus the Stokes number for monodisperse particles ranged from 0.85 ($d_p = 10$ μm, $V_\infty = 100$ m/s) to 27.28 ($d_p = 40$ μm, $V_\infty = 200$ m/s).

Since real particles are distributed in size, their behaviour in the carrier gas field is different. This results in particles redistribution in the flow. In calculations, the log-normal law with relatively small standard deviation $\sigma = \ln 1.2$ was taken for the particle size distribution and the most probable size r_m ranged from 10 to 40 μm (see Fig. 3).

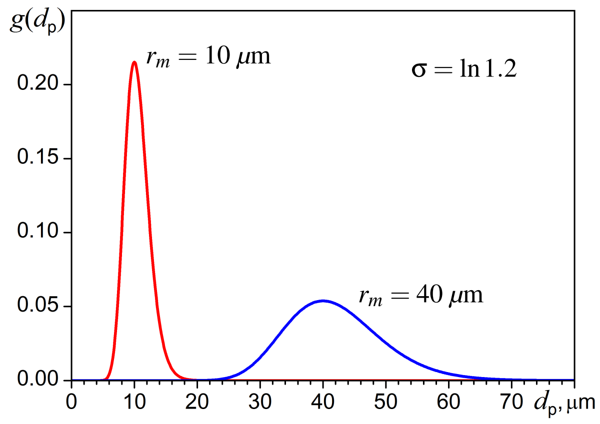

Fig. 3. Log-normal particle size distribution.

Being more inertial than the carrier gas, particles do not follow the streamlines and can collide with blades and rebound (reflect) from them.

We consider particle-blade collision as non-sliding and only partly elastic. In classical two-phase flow theory particles are assumed to have a spherical shape. The deterministic semiempirical particle-wall collision model [6] was applied for spherical particles. However, real particles (sand, ash, *etc.*) are not spherical, and because of this they are scattered in particle-wall collisions. We took ellipsoid of revolution and prism with cut vertices (see Fig. 4) as non-spherical particle shapes. The stochastic three-dimensional model [7] was used in simulation of particle-blade collisions for these particles.

The reflected particles can collide with the incident ones. These collisions, being random in nature, result in chaotic motion of the involved particles. A kinetic model of particle-phase flow with inter-particle collisions was proposed in [8]. It is based on considering the carrier gas as a continuum and the dispersed phase as a set of a large number of particles. The particle-phase flow is considered as three-dimensional and described by the kinetic Boltzmann equation generalized for the case when particles interact with the carrier gas and the particle-particle collisions are inelastic and frictional. The applicable DSMC algorithm was developed in [9] and it was adapted

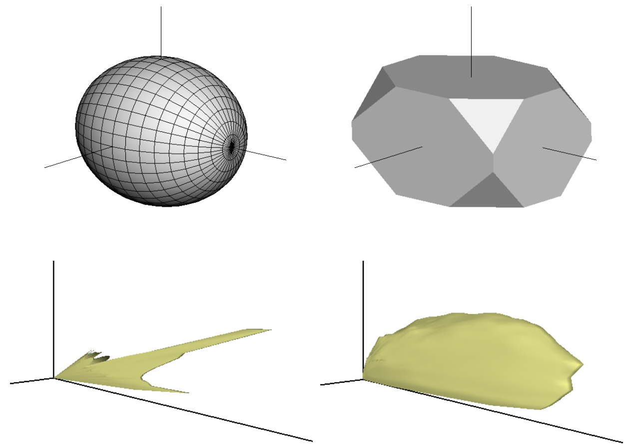

Fig. 4. Non-spherical particles used in simulation of particle-wall collisions and their respective scattering indicatrixes for collision angle of 15°.

for the present problem in [5].

In collisionless disperse phase flow particles move independently of each other, thus the Lagrangian trajectory approach was used.

5 Computational Results and Discussion

As an illustration of the considered effects, instantaneous patterns of particle cloud are shown for the flows with inlet velocity of 100 and 200 m/s with rather different structure of the carrier gas flow (see Fig. 2).

The influence of the carrier gas flow structure on the disperse phase behaviour is shown in Fig. 5. Particles are involved in large-scale vortex motion, whereas the effect of small-scale vortices is negligible due to inertia of particles. As it was mentioned above, the Navier–Stokes and URANS models result in different flow structure. This difference has reflection in particle-phase motion and is rather significant in the case of large-scale vortex flow (patterns *a*, *b*). Both flow models give very close results in the case of small-scale vortex wake (patterns *c*, *d*), because particle inertia makes inessential the flow fluctuations of rather small size as compared to the particle relaxation length. Hence, in multiphase flow simulations, only DNS or pseudo-DNS approach results in correct pattern of disperse phase flow when carrier gas flow structural features are substantial in terms of particles in-

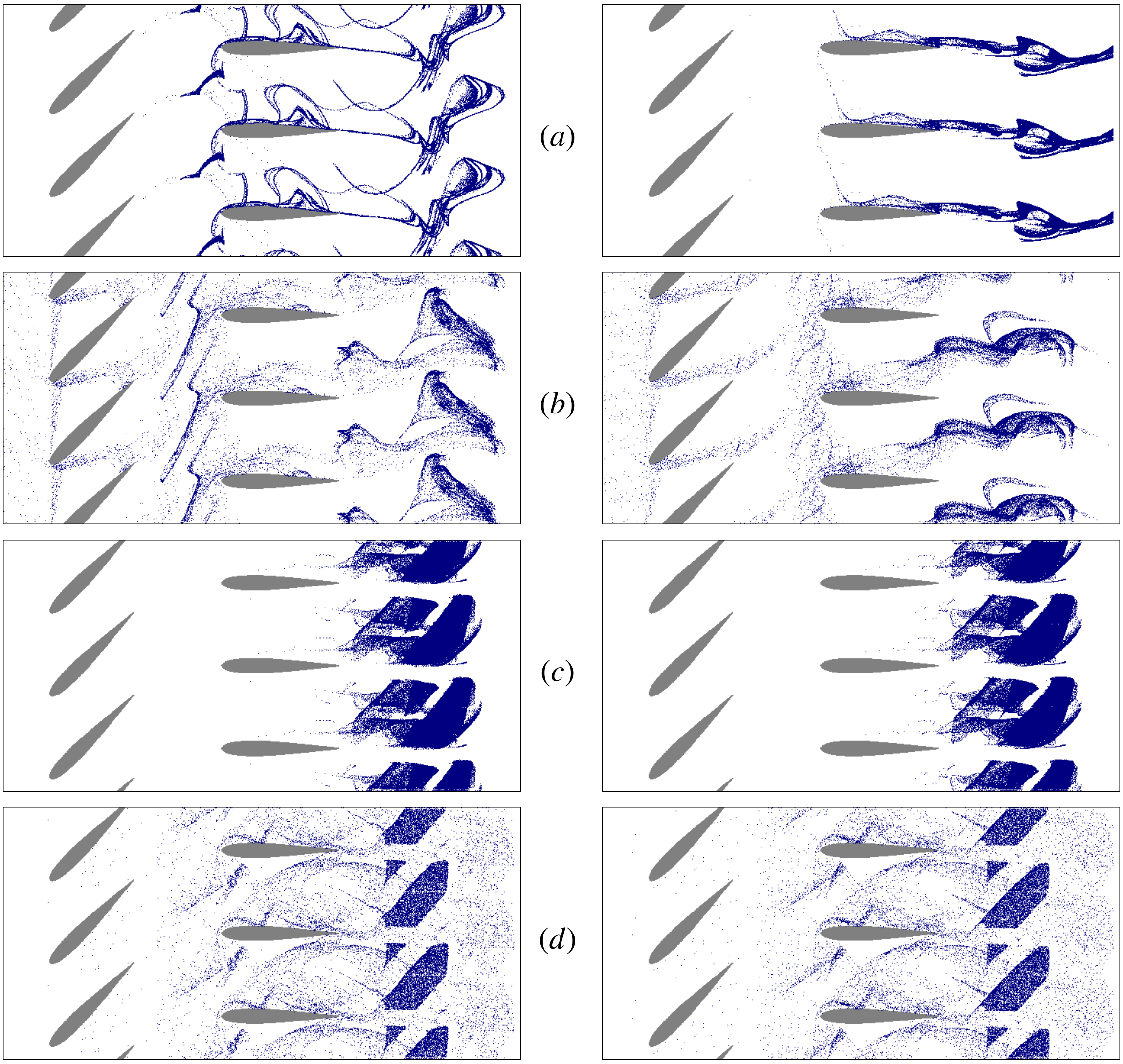

Fig. 5. Instantaneous patterns of particle cloud obtained using the Navier–Stokes flow model (left) and the $k-\omega$ SST turbulence model (right) for the flows with inlet velocity of 100 m/s (a, b) and 200 m/s (c, d), and spherical particles of 10 μm (a, c) and 40 μm (b, d) in diameter.

ertia. Even through some averaged parameters of disperse phase, such as particle concentration, are of interest, DNS-like approach should be used followed by a proper averaging procedure. Carrier gas flow model has no effect only when the flow structural features are small enough.

Although calculations were performed using the Navier–Stokes equations and the URANS approach with the Menter $k-\omega$ SST turbulence model, the results for particles are given below only for the Navier–Stokes flow model.

The effect of particle size distribution is shown in Fig. 6. It is seen that polydisperse particle clouds are more blurred. Particles are redistributed in carrier gas flow according to their size. Even for relatively small size dispersion considered, the ratio of the largest particle Stokes number to minimum one exceeds 20, that explains the obtained result.

The next essential factor which occurs in real flows is the non-spherical shape of particles. The effect of particle scattering in collisions with blades is illustrated in Fig. 7 very impressively when compared with Fig. 6. Upon a closer view, this effect depends on particle size, or more exactly on the particle Stokes number. It deserves consideration that cloud patterns for ellipsoidal and prismatic particles are too close, whereas

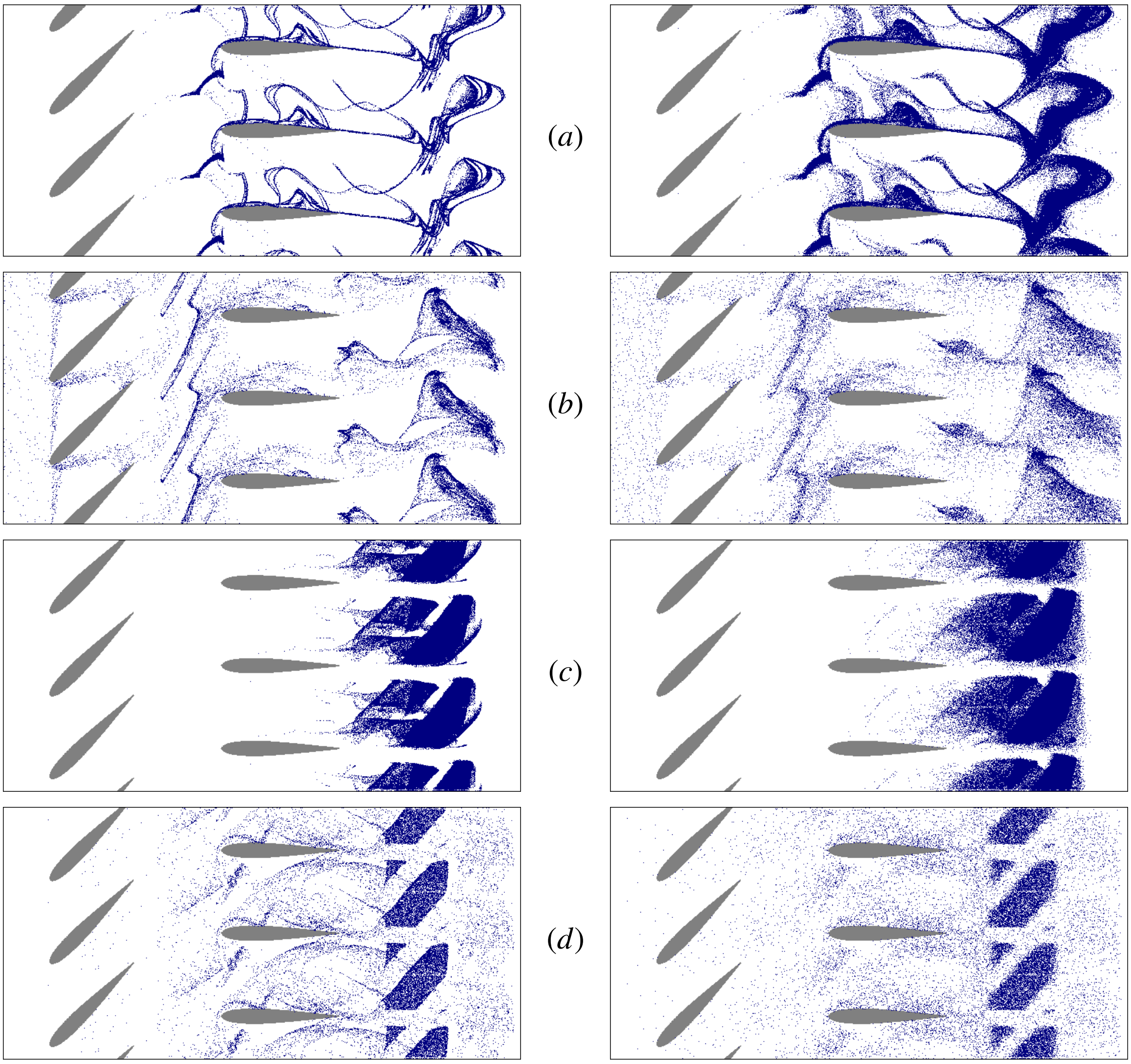

Fig. 6. Instantaneous patterns of monodisperse (left) and polydisperse (right) particle cloud in the flows with inlet velocity of 100 m/s (a, b) and 200 m/s (c, d), and spherical particles of 10 μm (a, c) and 40 μm (b, d) in diameter (fixed or most probable).

corresponding scattering indicatrixes are significantly different (see Fig. 4). For one thing, the particle scattering itself dominates over the particularities of the particle-wall collision. So, the differences in scattering of particles due to their shape or collision models have reflection in minor features of the particle-phase flow.

Both the particle size distribution and the scattering of non-spherical particles in collisions with blades are very important. These effects result in blurring of narrow layers with high particle concentration which arise in classical deterministic models of two-phase flow with spherical particles of the same size. On the one hand, such layers have been the focus of interest as potential cause of intensive local erosion of blades, but the obtained results show this problem to be overstated. On the other hand, considering size distribution of particles and their scattering in collisions allows to extend the validity range of the model of one-way coupled gas-particle flow without inter-particle collisions. It is illustrated by patterns of ellipsoidal particle clouds in Fig. 8 which are so close (compare left and right) without reference to account of inter-particle collisions.

Finally, the combined effect of particle size distribution and scattering is shown in Fig. 9.

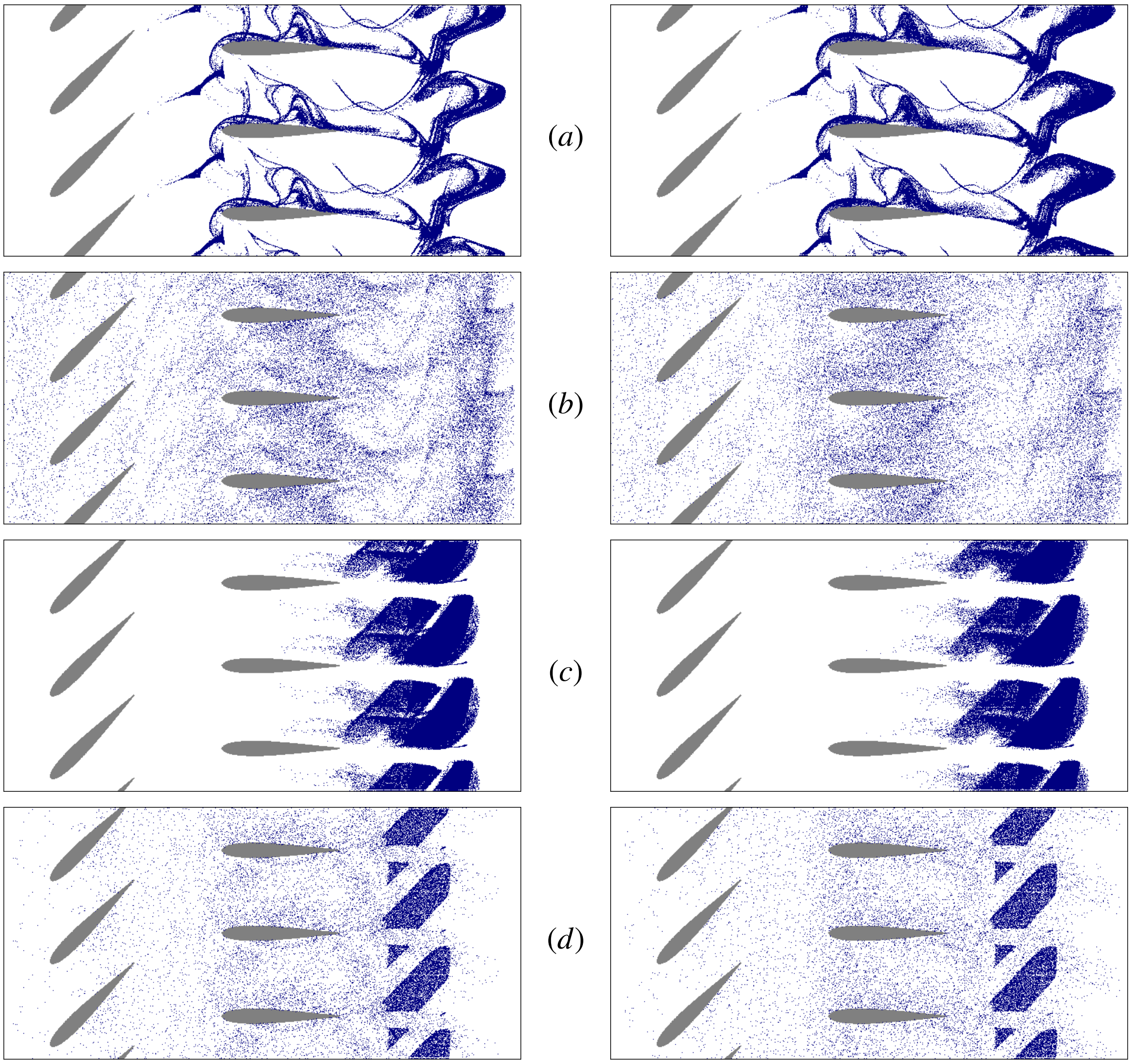

Fig. 7. Instantaneous patterns of particle cloud for ellipsoids (left) and prisms (right) in the flows with inlet velocity of 100 m/s (*a*, *b*) and 200 m/s (*c*, *d*), and particle size of 10 μm (*a*, *c*) and 40 μm (*b*, *d*).

6 Concluding Remarks

DNS-like carrier gas flow simulation is preferable when the particle relaxation length is small as compared to the linear scale of flow vortex structure. All the investigated random effects resulted in blurring of narrow layers with high particle concentration (such layers always appear in the case of mono-sized spherical particles). It was found that the particle size distribution and scattering of non-spherical particles play the most important role in redistribution of particles in the flow. The role of the particle-particle collisions turned out to be not as important as it was expected from estimations obtained for dusty gas flow over a forward part of a blunt body [10]. In simulation of particle motion, three-dimensional effects are important in particle scattering, collisions between particles, and also in some other cases [11], but they have very low effect on the particle flow structure in the plane of cascades.

Acknowledgements

The present work was supported by the Russian Foundation for Basic Research through grants Nos. 09-08-00888 and 12-08-01282.

References

[1] Hamed A, Tabakoff W and Wenglarz R. Erosion and deposition in turbomachinery. *Journal*

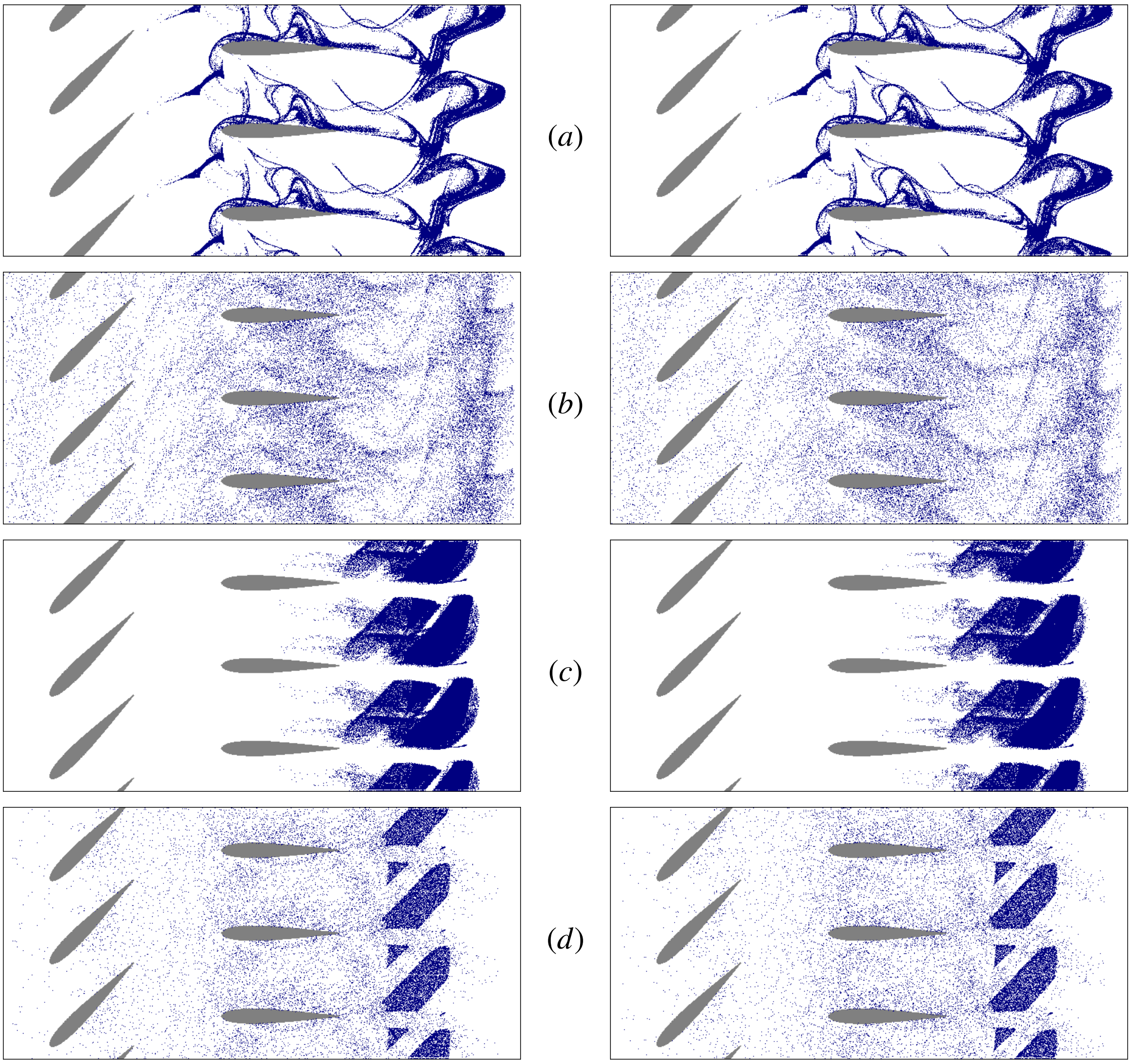

Fig. 8. Instantaneous patterns of particle cloud obtained without (left) or with (right) taking inter-particle collisions into account for ellipsoidal particles in the flows with inlet velocity of 100 m/s (*a*, *b*) and 200 m/s (*c*, *d*), and particle size of 10 μm (*a*, *c*) and 40 μm (*b*, *d*).

of Propulsion and Power, Vol. 22, No. 2, pp 350-360, 2006.

[2] Bons J.P. A review of surface roughness effect in gas turbines. *Journal of Turbomachinery*, Vol. 132, 2010. [DOI: 10.1115/1.3066315].

[3] Tsirkunov Yu.M., Romanyuk D.A. and Panfilov S.V. Effects of particle mixing and scattering in the dusty gas flow through moving and stationary cascades of airfoils. *Progress in Propulsion Physics*, Vol. 2, pp 459-474, 2011.

[4] Romanyuk D.A. and Tsirkunov Yu.M. Numerical simulation in unsteady dusty gas flow through the moving and stationary cascades of airfoils. *Proc. of 5th European Conference on Computational Fluid Dynamics – ECCOMAS CFD 2010*, Lisbon, Portugal, Paper No. 1063, pp 1-20, 2010.

[5] Tsirkunov Yu.M. and Romanyuk D.A. CFD/Monte Carlo simulation of dusty gas flow in a "rotor-stator" set of airfoil cascades. *Proc. of 5th European Conference for Aeronautics and Space Sciences – EUCASS'2013*, Munich, Germany, Paper No. 373, pp 1-12, 2013.

[6] Tsirkunov Yu.M., Panfilov S.V. and Klychnikov M.B. Semiempirical model of impact interaction of a disperse impurity particle with a surface in a gas suspension flow. *J. Engng. Phys. Thermophys*, Vol. 67, No. 5-6, pp 1018-1025, 1994.

[7] Panfilov S.V. and Tsirkunov Yu.M. Scattering

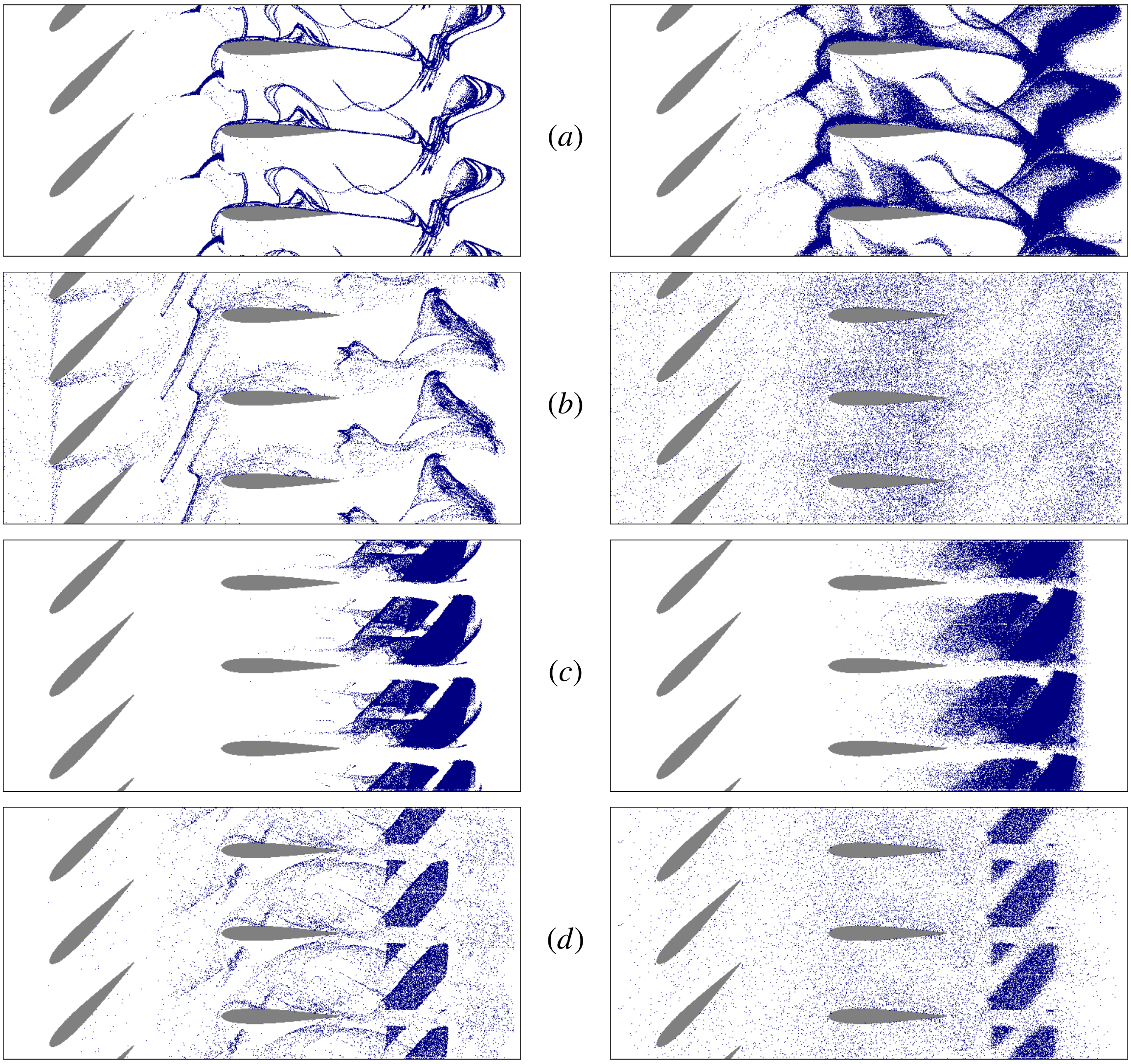

Fig. 9. Instantaneous patterns of monodisperse spherical (left) and polydisperse prismatic (right) particle cloud in the flows with inlet velocity of 100 m/s (*a*, *b*) and 200 m/s (*c*, *d*), and particle size of 10 μm (*a*, *c*) and 40 μm (*b*, *d*) (fixed or most probable).

of nonspherical particles rebounding from a smooth and a rough surface in a high-speed gas-particle flow. *Journal of Applied Mechanics and Technical Physics*. Vol. 49, No. 2, pp 222-230, 2008.

[8] Volkov A.N. and Tsirkunov Yu.M. Kinetic model of a collisional admixture in dusty gas and its application to calculating flow past bodies. *Izv. Russian Akad. Nauk, Mekh. Zhidk. Gasa*, No. 3, pp 81-97, 2000. (Engl. transl. *Fluid Dynamics*, Vol. 35, No. 3, pp 380-392, 2000.)

[9] Volkov A.N. and Tsirkunov Yu.M. CFD/Monte Carlo simulation of collision-dominated gas-particle flows over bodies. *Proc. of ASME FEDSM'2002*, Paper No. 31222, pp 1-14, 2002.

[10] Tsirkunov Yu.M. Gas-particle flows around bodies – key problems, modeling and numerical analysis. *Proc. of 4th Int.Conf. on Multiphase Flow – ICMF'2001*, New Orleans, LA, USA, Paper No. 607, pp 1-31, 2001.

[11] Hussein M.F. and Tabakoff W. Computation and plotting of solid particle flow in rotating cascades. *Computers and Fluids*, Vol. 2, No. 1, pp 1-15, 1974.

7 Contact Author Email Address

Yury Tsirkunov: tsrknv@bstu.spb.su

Copyright Statement