CFD / Monte Carlo simulation of dusty gas flow in a set "rotor–stator" of airfoil cascades

Yu.M. Tsirkunov and D.A. Romanyuk

Department of Plasma- and Gas Dynamics and Heat Engineering
Baltic State Technical University
1st Krasnoarmeiskaya str. 1, Saint Petersburg 190005, Russia
E-mail: tsrknv@bstu.spb.su

Military aircrafts can sometimes operate in dusty atmosphere that is illustrated by probably the most famous pictures from internet (see Figs 1 and 2). Civilian aircrafts can also fall within a cloud of volcanic ash. In all these cases, we deal with gas–particle flow inside engine duct. Dispersed particles suspended in a high speed inlet flow, being more inertial than the carrier gas, can collide with blades of a fan, a compressor, and even of a turbine resulting in abrasive erosion and quick wear of blades. That is why the problem of dusty gas flow in jet engines has been the focus of attention already for at least 40 years.

Fig. 1. Landing of an aircraft C130 "Hercules" in Afghanistan.

Fig. 2. Flight of F-15 and F-16 over burning oil wells during Gulf war in 1990–1991.

Modelling and simulation of dusty gas flow through the whole engine is very complex problem. Strictly speaking, the flow in the engine is time-dependent and three-dimensional, however many key flow features in the sequential rims of blades can be studied using a 2D flow model in the plane, which represents a developed mean-circle cross section. Such an approach is very fruitful and it is well known as the 2D theory of cascade flow. The present paper presents the results of computational simulation of time-dependent gas–particle flow through a set of two, moving and stationary, cascades of airfoils. The main aim of the study was to clarify the role of collisions between particles in redistribution of the dispersed phase in the flow. It is clear that the particle–particle collisions play no role if the particle volume fraction in the inlet flow α_∞ is vanishingly small, but if the particle concentration increases this role becomes noticeable already at very low α_∞ [1], and hence the collisions must be taken into account in this case. The following increase of α_∞ results in the noticeable effect of particles on the carrier gas flow. This effect is also the subject of the present study.

The mathematical two-phase gas–particle flow model is based on consideration of the carrier gas as a continuum and the dispersed phase as a set of a great number of particles the behaviour of which is 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. Such a model was proposed in [2]. Computer code for calculation of flow on the basis of this model implies CFD simulation of the carrier gas and Direct Simulation Monte Carlo of the particle phase. The applicable algorithm was developed in [3] and it was adapted for the present problem. As far as we know, such CFD/DSMC computational investigation of flow in a set “rotor–stator” of cascades has been undertaken for the first time.

In calculations, the particle size was varied from 2 to 40 μm, the inlet velocity $\boldsymbol{V}_{inlet}$ (see Fig. 3) was varied from the value corresponding to zero velocity in the undisturbed flow (engine runs at the air-strip) to the value corresponding to the cruise flight ($\boldsymbol{V}_\infty = 200$ m/s). The translational velocity of the rotor cascade was $\boldsymbol{V}_{rot} = 150$ m/s. Three flow models for the carrier gas were used: Euler equations, Navier–Stokes equations, and RANS equations with the Menter k–ω SST turbulent model. The semiempirical particle–blade collision model [4] was used. Schematic of flow is described in detail in [5]. Figure 3 illustrates the redistribution of particles (SiO_2) when an initially uniform particle cloud of finite thickness moves from left to right.

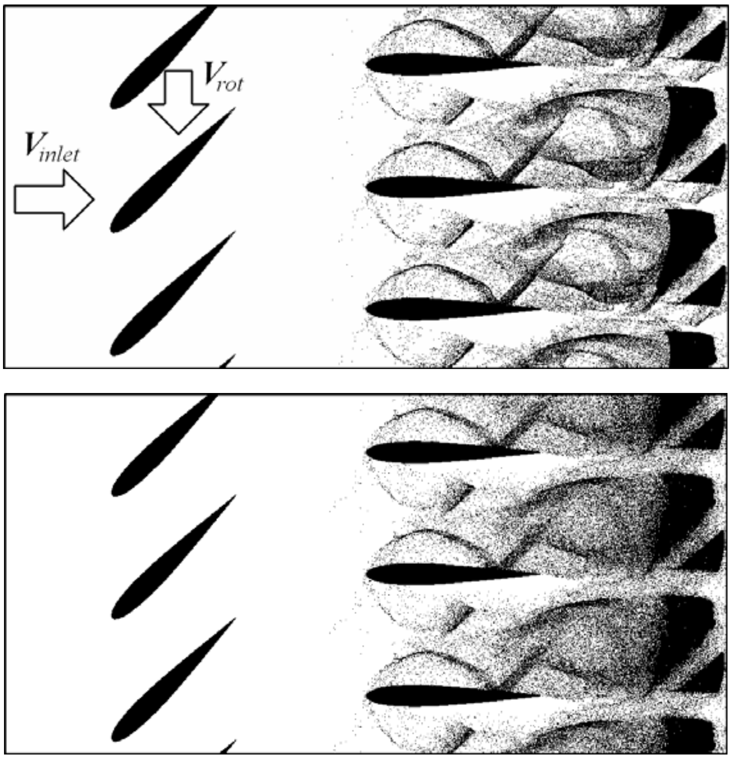

Fig. 3. Instant patterns of dispersed particles: simulation without (top) and with (bottom) collisions between particles. Particle diameter is $d_p = 20$ μm

As is seen, the particle–particle collisions results in smearing of sharply shaped layers with a high particle concentration. This effect becomes more pronounced as the particle size (more exactly, on the Stokes number) and the particle concentration increase. Similar smearing effects due to mixing of particles of different sizes and particle scattering in particle–blade collisions were obtained and described in our recent paper [5].

The present paper completes the study of random effects (particle mixing, scattering of reflected particles, and particle–particle collisions) in dusty gas flow though a set “rotor–stator” of airfoil cascades.

References

[1] Tsirkunov, Yu.M. (2001) Gas–particle flows around bodies – key problems, modeling and numerical analysis. *4th Int. Conf. on Multiphase Flow.,* Ed. E. Michaelides. New Orleans, LA, USA. CD-ROM ICMF’2001. Paper 607, 31 p. (invited lecture).

[2] Volkov, A.N. and Tsirkunov, Yu.M. (2000) 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, 81–97. (Engl. tranl. *Fluid Dynamics*. **35**, No. 3, 380–392.)

[3] Volkov, A.N. and Tsirkunov, Yu.M. (2002) CFD/Monte Carlo simulation of collision-dominated gas–particle flows over bodies. *Proc. of ASME FEDSM2002*, Paper No.31222, 14 p.

[4] Tsirkunov, Yu.M., Panfilov, S.V. and Klychnikov, M.B. (1994) Semiempirical model of impact interaction of a disperse impurity particle with a surface in a gas suspension flow. *J. Engng. Phys. Thermophys.* 67 (5-6), 1018–1025.

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