8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineeering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

DUSTY GAS FLOW THROUGH THE MOVING AND STATIONARY CASCADES OF AIRFOILS

*** Denis Romanyuk[1] and Yury Tsirkunov[1]**

[1] Department of Plasma- and Gasdynamics, Baltic State Technical University
1st Krasnoarmeiskaya str., 1, 190005 Saint Petersburg, Russia
E-mail address: tsrknv@bstu.spb.su

Key Words: *Two-Phase Gas-Particle Flow, Airfoil Cascades, Non-Stationary Flow, Computational Simulation.*

ABSTRACT

Flow through a cascade of airfoils occurs in aircraft turbojet engines, in gas turbines of power stations, *etc*. The conventional theory of cascades is based on a consideration of flow of a pure gas, whereas in practice, a working gas often contains suspended solid particles or liquid droplets. The presence of a dispersed phase in the flow results in some new effects which more often than not, are undesirable. Specifically, it causes the erosion of blades due to multiple impacts of particles or droplets with them and the additional momentum and energy losses [1]. The purpose of this work is to study the behaviour of solid particles in the time-dependent high-speed subsonic 2D-flow through a set of two, moving and stationary, airfoil cascades.

A dusty gas flow with a very low particle mass load is considered. The inter-particle collisions and the effect of the dispersed phase on the carrier gas flow are assumed to be negligible. In this case, the problem of two-phase flow simulation can be reduced to the sequential solving of two problems: (i) computation of the carrier gas flow field, and (ii) calculation of the particles' motion in this flow field.

Flow of a compressible gas through the set of cascades is described by the complete Navier–Stokes equations. In every cascade, a problem for a single airfoil is considered. The whole flow field is obtained by periodic repetition of a single airfoil solution within the moving or stationary cascade, respectively, with simultaneous matching the solutions between both cascades.

The computational domain is composed of two blocks the first of which moves relative to the second one. The boundary of every block consists of the contour of an airfoil and the outer contour which has the form of a parallelogram. Both blocks had a common contact line parallel to the cascades. At the airfoil surface, the no-slip condition and the constant-temperature wall condition are enforced. The periodic boundary conditions are used at the top and bottom outer boundaries. At the inflow boundary of the moving block, the stagnation enthalpy h_0 and the entropy function $\vartheta = p/\rho^\gamma$ are given, and the static pressure p_{in} is extrapolated from the calculational domain. At the outflow boundary of the stationary block, the pressure p_{out} was taken equal to $1.2\,p_{in}$ that corresponded to some experimental data. At the contact line between the moving and stationary blocks, a matching procedure is used.

A curvilinear boundary-fitted non-orthogonal grids refining towards the airfoil contours were introduced in every block. The Navier–Stokes equations were solved using a finite-volume method of the second order [2]. To estimate the "numerical diffusion" of the method, the results were compared with those obtained from the Euler equations with the use of the same grid.

In computations of a dispersed phase flow pattern, particles were assumed to be spherical and of equal diameter. In the model of the gas–particle interaction, the drag force, the lift Magnus force and the damping torque were taken into account. The last two factors play an important role when particles acquire high rotational velocity in the process of particle–blade impact interaction. The parameters of a particle just after its rebound from the blade surface were calculated with the use of a semi-empirical particle–wall collision model. Further details and references are given in [3].

The momentum and angular momentum equations for a single particle together with the kinematic relations between the particle coordinates and the velocity components are solved numerically. If any particle crosses the top or bottom boundaries of a block, it is excluded from calculations, and a new particle with the same parameters is introduced into the same block at the corresponding point of the opposite boundary. An undisturbed gas–particle flow upstream from the first (moving) cascade is assumed to be uniform with equal velocities of both phases. Particles crossed the outflow boundary are excluded from further calculations.

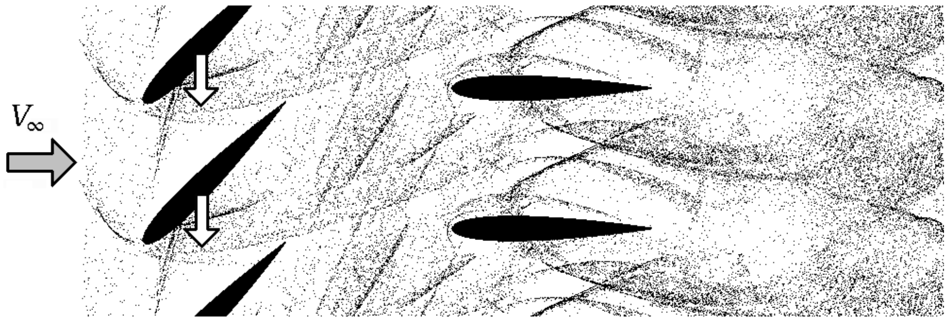

Figure 1: Example of instant pattern of particles moving through the cascades of airfoils.

The input data in computations (flow properties, speed of a moving cascade, airfoil sizes, *etc.*) were taken close to those in the flow through an axial compressor of an aircraft turbojet engine. For visualization of the particle-phase flow, a particle cloud of finite width was considered in an undisturbed flow. An example of instant pattern of particles from this cloud in both, moving and stationary, cascades is shown in the figure. Thin layers with high particle concentration being formed in the flow and moving with high speed can present a severe hazard to blades because of the erosive action. In the paper, flow patterns for particles of different sizes are discussed and analyzed.

This study was supported by the Russian Foundation for Basic Research (Grant No. 05-08-50075).

REFERENCES

[1] M.F. Hussein and W. Tabakoff. "Dynamic behavior of solid particles suspended by polluted flow in a turbine stage". *J. Aircraft*, Vol. **10**, No. 7, 543–561, 1993.

[2] C. Hirsch. *"Numerical computation of internal and external flows"*. Vols. **1** and **2**. Wiley, Chichester, 1994.

[3] Yu.M. Tsirkunov, S.V. Panfilov. "Particles scattering in particle-wall collisions and its effect on the particle-phase flow". *Proc. EUROMECH Colloquium 447: Proc. Estonian Acad. Sci. Eng.* Vol. **11**, No. 2, 126–139, 2005.