A Numerical Simulation of Particle Laden Flow through Airfoil Cascade

Stanislav Ivanov

Abstract: *Particle laden flows in multi-connected areas are subject of practical interest. Direct numerical simulations of such flows are limited by the size of needed computational resources. The possibilities of a simple computational model are discussed in this paper. A two step numerical simulation is proposed as a way to extend the applicability of this model. The proposed steps of simulation are: calculation of trajectories of particles with selected size and data acquisition for velocity and density fields for different particle sizes; generation of impacts between particles of different size and calculation of particle trajectories with simulated inter-particle collisions.*

Key words: *Numerical simulation, multi-phase flow, particles interaction, collision.*

INTRODUCTION

Flows of fluids carrying dispersed solid particles are subject of interest in many technological areas: air pollution, duct transport etc. The work process in gas turbines and pump impellers is an instance of flow in multi-connected area. Such a flow can be laden by particles, like dust, sand, pulverized materials. In this case, the possible damage of blades due to mechanical erosion by particle impacts is the reason for many studies [1]-[3]. To predict the zones of erosion, moreover, to design a cascade with reduced erosion are engineering problems which lead to huge size computational models. Generally, the multi-phase multi-component flows are subject of computational hydrodynamics. Grid numerical methods are used preferably, taking into account various combinations of physical effects like fluid-particle interaction (one- or two-way), external forces acting on particles, collision models (particle-wall and particle-particle) etc. The complexity of the flow area and the variability of conditions about particles are obstacles for this approach in the case of blade cascade. Another approach is to analyze directly the particle system movement by solving the corresponding system of ordinary differential equations. The only trouble here is the size of such a system. With the aim to represent particle-to-particle collisions in the cases of interest, the size of this system should be of the order of 10^5 or higher. This paper presents an approach to compose a simulation model with realistic representation of dominating physical phenomena and acceptable waste of computational resources. The original problem is to explore the work of a pump impeller which transports mixture of water and sand with various granulometric compositions. The simulation model should take into account particle-to-wall and particle-to-particle impacts.

BASIC ASSUMPTIONS

The above-formulated problem was solved on the base of the simplest model of particle-flow motion – an isolated particle is carried by a steady flow [4]. The use of this model as a primary one is motivated by the following assumptions which take place in the case under consideration.

- The fluid flow is steady potential one;
- The only external force is the gravity;
- The presence of particles does not affect the flow;
- The particles enter the cascade area with constant velocity;
- The form and the rotational movements of particles can be neglected.

The underlying physical considerations for the assumptions above are: potential flow represents well the mean parameters of the real turbulent flow; the particles concentration is low; particles acquire constant velocity in a large inlet area; the modelling aims to reach a generalized picture of the process.

These assumptions allow the hypothesis that particles of equal size don't collide before possible impact to the walls and reflection. Trajectories of such particles can be gathered one-by-one. This is the reason such model (so called "particle in flow" model) to be widely used for exploring flows of monodispersed mixtures.

SIMULATION STAGES

As inter-particle collisions take place mainly between particles of different size, a simulation of particle laden flow with particle-particle and particle-wall collisions can be compiled as follows:

1. Solution for the main fluid flow;
2. Selection of representative particle sizes;
3. Calculation and recording of trajectories for each selected size, so that impacts with the walls to be detected and the whole inter-blade area to be covered;
4. Generation of impacts between particles of different size on the base of probability, scaled by their volume concentration and relative velocity;
5. Tracing the trajectories after impacts.

The first three steps compose the method "particle-in-flow", the next two – the new proposal for its further development. Besides the simplicity and efficiency of the primary method, two main advantages of this proposal can be outlined:

- as numerous pictures of particle flows are gathered executing step 3, the critical about erosion sizes can be easily recognized;
- once these pictures are collected, multiple simulations of particle collisions can be produced using different granulometric compositions and concentrations.

PROJECT SOLUTIONS

1. Main flow solver

The flow through a pump impeller is calculated using a set of flat sections. Each of them after conformal transformation represents an infinite periodic cascade of congruent airfoils. The flow is two-dimensional in the infinite area outside the airfoils. As a method for solving such a flow, the boundary elements method is chosen. This is one developed by L.Panov as described in [5]. After finding the vortex distribution over the contour of a chosen airfoil and adding the induction of the remaining cascade, the fluid velocity at every point of the area can be calculated. A new program implementation was developed, taking special care to ensure precise calculation of the flow parameters over the whole area. The picture of solution (fig.1) shows that streamlines obey well the periodicity no matter where they initiate.

2. Trajectories and streamlines solver

A particle trajectory can be found solving initial problem for a system of ordinary differential equations. Having in mind that trajectories can be broken by collisions, one-step method should be applied. With the reason to ensure high precision, a method of 5th order (Kutta-Merson) is implemented. The streamlines are not necessary for the simulation, but are a good test for main flow and trajectories solvers altogether.

3. Forces and impact modeling

Taking into account the forces, produced by pressure, drag, gravity and buoyancy, the following system of differential equations is composed for solving the trajectories.

$$\begin{aligned} \frac{d\mathbf{r}}{dt} &= \mathbf{v} \\ \frac{d\mathbf{v}}{dt} &= \lambda.Fr.|\mathbf{c}|.grad\,|\mathbf{c}| + \frac{18.\lambda.\zeta(\mathrm{Re}).Fr}{d.\mathrm{Re}}(\mathbf{c}-\mathbf{v}) + (1-\lambda)\mathbf{g} \end{aligned} \qquad (1)$$

All values are dimensionless and mean as follows:

- vectors $\mathbf{r}, \mathbf{v}, \mathbf{c}, \mathbf{g}$ are respectively the particle position and velocity, the fluid velocity at the particle position and the gravity unit vector;
- $\lambda, \zeta(\mathrm{Re})$ are correspondingly the fluid-particle density ratio and drag function;

- the similarity criteria are those of Reynolds and Froude and the dimensionless particle size: Re, Fr, d .

The initial (Cauchy's) problem for the trajectories is set giving values of the particle position far before the cascade front line and of the particle velocity such that the drag force, gravity and buoyancy balance to zero. In other words, initial particle velocity ensures that the second equation of (1) has 0 at its right.

The particle impact to the blade wall is represented as a pure elastic shock, producing reflection from the wall as follows:

$$\mathbf{v}' = \mathbf{v} - \mathbf{n}.(\mathbf{n} \bullet \mathbf{v}).2 \tag{2}$$

$\mathbf{v}$ and $\mathbf{v}'$ are the particle velocity before and after impact; $\mathbf{n}$ is the normal unit vector of the wall and $\bullet$ denotes dot product of vectors.

The particle-to-particle impact changes velocities of colliding particles i and k according to the impulse conservation law:

$$\mathbf{v}'_i = \mathbf{v}_i + \frac{m_i + m_k}{2m_i}(\mathbf{v}_k - \mathbf{v}_i) \tag{3}$$

where ' denotes "after impact", *m* can be substituted by the cube of corresponding particle size and the subscripts are interchangeable.

PROJECT DESIGN

The simulation itself and all preliminary calculations are developed as C++ project with the reasons of efficiency and code maintenance. A brief description of the classes and their purpose follows, related to the main steps of proposed simulation.

1. Main flow solution and fluid velocities
- class Blade: geometry of the airfoil contour and the cascade, check whether given point belongs to the flow;
- class VortSeg: line segment with vortex distribution on it – used to calculate induced values of the stream function and fluid velocities;
- class BaseFlow: defines abstract functions to describe the basic stream: stream function and velocity;
- class Gauss: provides solution of a system of linear equations (used by FlowSolver);
- class FlowSolver: using Blade and a derivative of BaseFlow defines the boundary problem and then solve it, producing the vortex distribution over the blade contour;
- class Flow: generated by FlowSolver. Given a point, checks the point and if correct, returns fluid velocity and its derivatives;

2. Trajectories calculation
- class VectorN: defines operations with vectors of n+1 numbers (double).
- class Merson: provides a function to execute one step of Kutta-Merson integration method using VectorN;
- class Trace: defines the initial problem for a separate trajectory and calculate it using Merson and Flow;

3. Trajectories registration
- class Mesh: creates an orthogonal mesh that cover the channel between two neighbor airfoils using streamlines and equipotential lines as curvilinear coordinates; for each node of the mesh mean values of the density and velocity can be stored; when a particle trajectory enters a cell of the mesh these values are updated;
- class TrajFile: maintains data for calculated trajectories;
- class FiltReg: executes the tracing of multiple trajectories so that inter-blade area to be covered, accepts the results from Trace, registers them into TrajFile and Mesh;

4. Particle impacts simulation
- class ImpGen: given two registered sets of trajectories generates an inter-particle collision and traces both trajectories after that;
- class Simul: contains functions for interactive control of the simulation and other services;

RESULTS

The initial goal of the solution was to check the initial hypothesis and to gather pictures of the flow of different sizes. It was found that:
- having unified initial conditions, the particle trajectories really don't intersect;
- particles with a size of 0.005 and smaller have trajectories very close to the streamlines.
- trajectories which can produce inter-particle impacts emerge in a narrow interval of sizes

At figures 2, 3 and 4 trajectories for the size *d*=0.01, 0.015, 0.018 are shown . The other parameters are $\lambda = 0.5, \mathrm{Re} = 20000.d, Fr = 0.5, \zeta(\mathrm{Re}) = 1.$

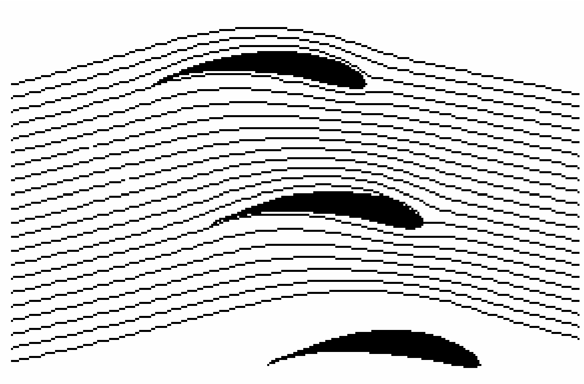

Fig. 1

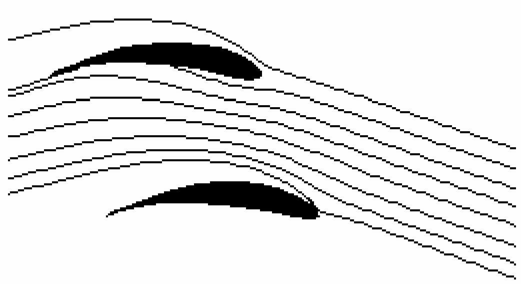

Fig.2

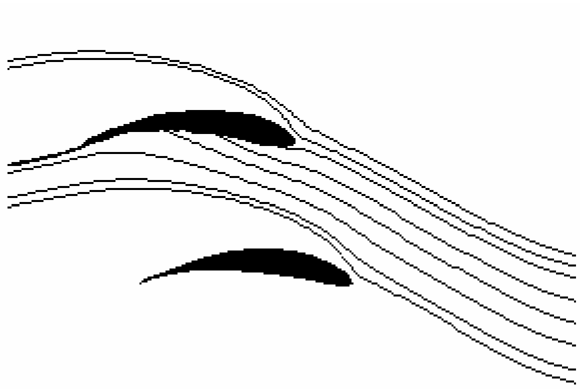

Fig. 3

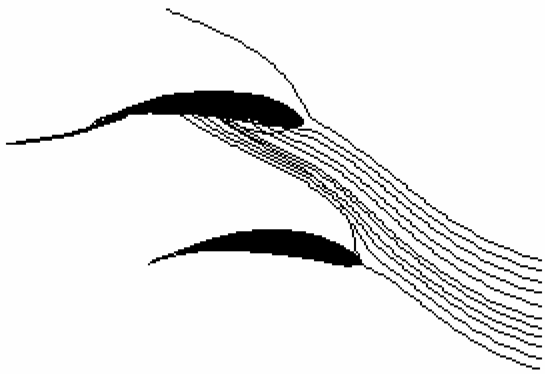

Fig.4

CONCLUSIONS AND FUTURE WORK

This work had been initiated with the hidden idea that something like particle velocity field can be calculated and used. The implementation of the model shows that this idea is useful. It was found during the computational experiment that in the flow can exist zones of concentration and zones of absence of particles of given size. Actually, the presence of these zones substantiates the two step simulation proposed in this paper. The results so far show that the cases of practical interest (small size particles) can be explored without such a simulation. But these results have been produced under simplified physical model in order to test the basic idea and the application. This work would be the ground for exploring the real process in pump impellers in three dimensions and using more precise model of some effects, e.g. collisions with friction and rotation.

REFERENCES

[1] Hamed, A., Effect of Particle Characteristic on Trajectories and Blade Impact Patterns, J. of Fluid Engineering, Vol. 110, pp. 33-36, 1988.

[2] Metwally, M., W. Tabakoff, A. Hamed, Blade Erosion in Automotive Gas Turbine Engine, Trans. ASME, J. of Engineering for Gas Turbines and Power, Vol. 117, pp. 213-219, 1995.

[3] Cho H., W. S. Kim, M. S. Yu, J. C. Bae, Analysis of Particle Laden Flow and Heat Transfer in Cascade and Rocket Nozzle, Annals of the New York Academy of Sciences 934:233-240, 2001

[4] Ivanov S., R.Andreev, I.Belberov , M.Senden , Simulation Modeling of a Solid Particle Motion in Zigzag Air Classifiers, ASTIC Proceedings, Varna, 1984

[5] Попов М., Л.Панов, Хидро- и газодинамика, Техника, 1980.

ABOUT THE AUTHOR

Assoc.Prof. Stanislav Ivanov, PhD, Department of Computer Science, New Bulgarian University, Phone: +359 2 811 0611, E-mail: sivanov@nbu.bg.