LARGE-EDDY SIMULATION OF PARTICLE DISPERSION IN THE DUCT WITH FLUID INJECTION

Konstantin N. Volkov*

University of Central Lancashire, Harris Building, Corporation Street, PR1 2HE Preston, UK

Summary Turbulent gas-particle flows in the ducts with fluid injection are simulated using large eddy simulation technique. The constructed computational approach is taking into account the velocity and temperature fluctuations of turbulent duct flow. The influence of inflow conditions, size of particles and place of particles injection on their dispersion pattern in the duct flow are investigated.

INTRODUCTION

Gas-particle flow research has applications to a vast number of technological systems and industrial devices. In technological applications the interaction of the particles with the turbulent structure of the flow is an extremely complex problem. The introduction of new models and improved computer power allows making more accurate computations performed with less empiricism than before. The technique is supposed to make possible studying the physical phenomena that occur in more complex engineering-like applications. Large eddy simulation (LES) methodology corresponds to those purposes.

The application of LES for computation of turbulent flows in the ducts with fluid injection is considered in this paper. Simulation of such flows is of a great significance for the designing and perfection of heat exchangers and solid rocket motors. In these applications exact computation of field of fluid flow plays an important role because it serves as a background for simulation of condensed particles formed during the combustion of solid fuel, their interaction and separation on a wall [1, 2]. The computational domain is shown in the figure 1. The purpose of established calculations is definition of optimal initial parameters of duct flow and lead-in conditions of solid particles.

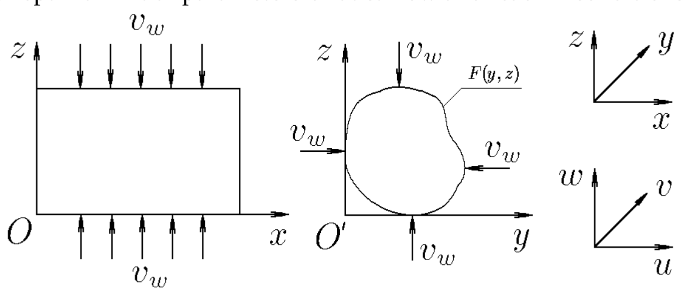

Figure 1. Computational domain

MATHEMATICAL FORMULATION

The numerical calculations are performed by the Eulerian-Lagrangian approach for the fluid and dispersed phases respectively.

Gas phase

The fluid flow calculations are based on filtered Navier-Stokes equations. Subgrid turbulent viscosity is computed with the Smagorinsky's formula and the dynamic model is used for estimating the Smagorinsky's parameter. The subgrid length is assumed to be proportional to the filter width, which then is related to the volume of grid mesh in three dimensions. The present LES implicity applies a box filter in each direction.

Dispersed phase

The solid phase is treated by the Lagrangian approach, which means that particles are followed in time along their trajectories through the flow field. At every given time step, the new position of the particles and the new translational velocity are calculated according to the forces acting on the particles. LES allows obtaining fluctuating components spontaneously from momentum equation unlike approach [1, 3] which uses normal distribution for fluctuating properties of turbulent fluid flow. It is assumed that the particles are undeformable spheres, there is no particle-particle or droplet-droplet interaction, density of dispersed phase is much bigger than density of gaseous phase, the dominant forces acting on the particle are the drag, influence of particle phase on carrying gas flow is negligible.

NUMERICAL METHOD

The filtered equations are solved numerically using a finite volume method on staggered grid with non-uniform cell size. The numerical procedure employs a Chorin-type projection method for the decoupling of momentum and

continuity equations. The second order explicit Adams-Bashforth time integration scheme is used to advance the velocity field. The discretization of diffusive flux is based on the central differences of second order. Concerning numerical method, the most important component is the discretization of the non-linear convective fluxes. The sharp and monotonic algorithm for relative transport (SMART) is used [4]. The Poisson equation for pressure is solved by the bi-conjugate gradients stabilised method (BiCGStab) with preconditioning [5].
The computational domain is divided into several rectangular subdomains for the parallelization. Each process holds some ghost cells, which overlap inner cells of the adjacent process. Values are copied from these to the ghost cells when necessary. To minimise communications, the program divides computational domain in a way that minimises the area of the touching faces and equilibrates the number of cells in the different subdomains. The MPI library is used for organisation of communication between processors.

RESULTS

The presented calculations have been performed for demonstrating the influence of the different physical phenomena on the development of the particles dispersion. In order to underline the particles motion exceptions in the duct, these calculations were performed only for one-way coupling approach.
The grid containing 100 000 cells in the cross section of duct and 300 sections downstream was used in typical variant of computation. Some problematic issues concerning the methodological base of LES were investigated. The main results of the paper are the following: (i) elaboration of convergence criteria of LES (in other words, how filter length influences computational results and how filter kernel influences treatment of computational results); (ii) development of numerical schemes ensuring necessary accuracy; (iii) development of LES for computation of turbulent flows on the basis of general curvilinear grid structures; (iv) development of parallel numerical algorithm for high performance computer systems both with shared and distributed memory.
Fluid flow pattern was investigated depending on intensity of fluid injection. The formation of recycling regions and reverse flow in the duct was discovered with the some of initial data. The computed results are compared with the approximate solutions, the available benchmark solutions and experimental data.
The dispersion of particle is defined by the relation of the particle relaxation time to the dissipation dynamic time scale of the turbulence. The particles with radius $r_p<10$ μm hit a target in the region with a maximum value of fluid velocity turbulent fluctuations. Such particles are dispersed strongly. The region of increased particles concentration is formed in the vicinity of the centerline of the duct. Inert particles penetrate to the contrary wall, where turbulence level is equal to zero, and are not dispersed.
The obtained results have a good agreement with the results computed on the base of Reynolds averaged Navier-Stokes equations and experimental data [5].

CONCLUSIONS

The simulation of transition processes of momentum and heat in the duct with fluid injection shows that the fluctuations of turbulent flow exercise an essential influence on dispersion and heating of particles. The developed model and obtained results reasonably explain computational and experimental data. In particular, they explain the formation of regions of irregular particles concentration in the turbulent two-phase duct flows.
Successful application of LES for computation of fluid properties in the duct with fluid injection makes possible computations taking into account more complex physical and chemical effects. It requires development subgrid scale models for compressible fluid, connection combustion model to LES concept and elaboration of corresponding computational tools.

References

[1] Volkov K.N., Emelyanov V.N.: Stochastical model of condensed particle motion in the duct with fluid injection. *Mathematical simulation* **11**(3):105-111, 1999.
[2] Volkov K.N., Emelyanov V.N.: Three-dimensional flows in the ducts with fluid injection. *Mordern Problems of Non-Equilibrium Gas and Thermodynamics* 43-63, 2002.
[3] Gosman AD, Ioannides E.: Aspects of computer simulation of liquid-fueled combustors. *AIAA Paper* 81-0323.
[4] Choi S.K., Nam H.Y., Cho M.: A comparison of higher-order bounded convection schemes. *Computational Methods in Applied Mechanics Engineering* **121**:281-301, 1995.
[5] Van der Vorst H.A.: Bi-CGSTAB: A fast and smoothly conversing variant of Bi-CG for the solution of non-symmetric linear systems. *SIAM Journal of Science Statistical Computations* **13**:631-644, 1992.