31. SETKÁNÍ KATEDER MECHANIKY TEKUTIN A TERMOMECHANIKY
26. – 28. června 2012, Mikulov

Turbulence modulation in Large Eddy Simulation of backward-facing step flow laden with particles

Jaroslav Volavý[1], Miroslav Jícha[2]

[1] Vysoké učení technické v Brně, Fakulta strojního inženýrství, Energetický ústav,
Technická 2896/2, Brno, yvolav00@stud.fme.vutbr.cz

[2] Vysoké učení technické v Brně, Fakulta strojního inženýrství, Ústav konstruování,
Technická 2896/2, Brno, jicha@fme.vutbr.cz

Abstract This work describes Large Eddy Simulation of backward-facing step flow laden with particles. The concentration of the particles in the flow is high enough for consideration of two-way coupling. This means that the particles are influenced by fluid and vice versa. The inter-particle collisions are neglected. The Euler-Lagrange method is adopted, which means that the fluid is considered to be continuum (Euler approach) and for each individual particle is solved Lagrangian equation of motion. Particles are considered to be spherical. The simulations are performed for different volume fractions and various types of particles. The results are compared to the single-phase flow in order to investigate the effect of the particles on the turbulence statistics of the carrier phase.

1 Introduction

In many industrial areas we can encounter the processes in which the particles transportation by turbulent flow is involved. Therefore there is an effort for accurate prediction of the particles motion and their deposition. Many forces act on the particles, for example the drag and lift force which is caused by surrounding fluid or gravity force. It is very important to resolve the flow and turbulence of the carrier phase in order to determine movement of disperse phase (particles) with sufficient precision because this affects transportation and deposition of particles most. In recent years the Large Eddy Simulation method was improved so far that it became capable of very accurate prediction of the turbulent flow and could be used for particle-laden flows.

In this work we studied the influence of particle on carrier phase and the modification due the presence of particles on the fluid. For solution of the carrier phase was used Large Eddy Simulation method, dispersed phase is simulated using the Lagrangian approach. The simulations were carried out in the OpenFOAM, which is freely available and accessible to the source code. The results of the simulation are compared with experimental results published in [1].

2 Mathematical model

For the solution of the fluid flow in this article was chosen Large Eddy Simulation. The main idea of Large Eddy Simulation is to separate large scales (grid-scales) from small scales (subgrid-scales) to lower computational cost. The subgrid scales are modeled using subgrid model. The scale separation is done by applying filter operator on Navier-Stokes equation. If we apply the filter operator on Navier-Stokes equations we obtain filtered Navier-Stokes equations [2]:

$$\frac{\partial \bar{u}_i}{\partial t}+\frac{\partial \bar{u}_j\bar{u}_i}{\partial x_j}+\frac{\partial \bar{p}}{\partial x_i}-\frac{1}{Re}\frac{\partial^2 \bar{u}_i}{\partial x_k \partial x_k}=\frac{\partial \tau_{ij}}{\partial x_j}-f_i, \tag{1}$$

where f_i is force that includes the effect of particle on the fluid phase. It is defined as

$$f_i = \sum_{n=1}^{N} f_{n,i} \tag{2}$$

$f_{n,i}$ is i-th component of the force acting on the n-th particle. Force $f_{n,I}$ is equal to the rand hand side of equation (3). The subgrid stress tensor was evaluated using Smagorinsky model.

The motion of the particles is described by Lagrangian equations of motion. The only force acting on the particles is drag force. All other forces were neglected. Standard drag correlation was applied. The equation of motion for j-th particle has form [3]:

$$\frac{dv_j}{dt} = \frac{u(x_j,t) - v_j}{\tau_p}\left(1 + 0.15 Re_p^{0.687}\right) \tag{3}$$

$\mathbf{v}_j$ is the particle velocity, $\mathbf{u}(\mathbf{x}_j,t)$ is velocity of the fluid seen by particle in position $\mathbf{x}_j$ and $\tau_p = \rho_p d_p^2 / (18 \rho_f \nu)$ je relaxation time of particle.

3 Description of the test case

Backward-facing step with particles was chosen as a test case. This case provides an ideal combination of flow in terms of simple geometry, on the one hand and the relatively complex nature of the flow on the other. The geometric configuration, flow parameters and boundary conditions based on the experiment described in [1]. The channel is positioned vertically, the acceleration of gravity acts in the direction of flow. Inlet channel height is 40 mm, which then abruptly expands to 66.7 mm. Hence expansion factor is 5/3.

The ratio of the width of the domain, which was conducted real experiment, and the step height was 17:1. This option reduces the influence of the side walls and provides two-dimensional character of turbulence statistics in a large part of the domain. In terms of calculation, however, this option represents a complication in the form of a rise in the number of grid cells, thereby increasing computing demands. Therefore, it was only simulated 120 mm slot width and periodic boundary condition was prescribed on the sides.

The computational mesh consists of 2.1 million hexahedral cells. The mesh is block-structured and becomes finer towards the wall in order to meet condition of $y^+ \approx 1$. The detail of the mesh near the trailing edge is in the **Fig. 1**.

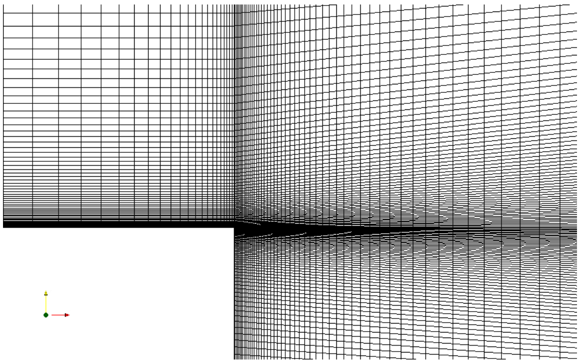

Fig. 1 Detail of computational mesh

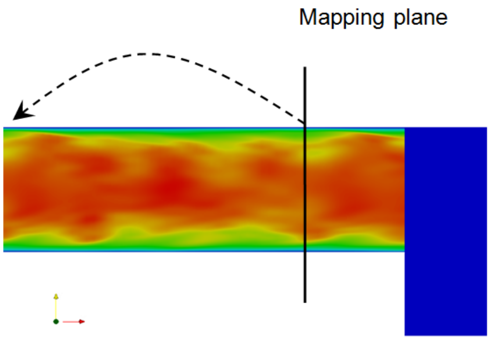

Fig. 2 Scheme of inlet data mapping

Reynolds number based on the inlet channel bulk velocity and height is 13,800. The fluid is air. On the inlet is supposed to be fully developed turbulent flow. This was achieved as follows.

First, the simulation of the inlet channel only was performed with periodic boundary conditions in the direction of the flow and on the sides. To accelerate the transition to turbulence excitation scheme based on Ornstein-Uhlenbeck process was used [4]. When a fully developed turbulent flow regime was achieved, the results were mapped to the backward-facing step geometry. A perpendicular plane was defined at a distance of 80 mm from the entry. The velocity on the inlet is obtained by direct mapping of the velocity from this plane. This procedure is schematically illustrated in the **Fig 2**. The simulation ran 0.5 s in order to develop turbulent structures in the rest of the domain.

The glass particles with diameter of 150 μm and density 2500 kg.m^{-3} were chosen as dispersed phase. The particle diameter is greater than the lowest dimension of the computational cell therefore it is appropriate to use above mentioned model. The mass loading of the particles in the flow is 20% and 40%.

The end state of the simulation without particles was used as an initial condition for the simulation with particles. Particles were uniformly distributed over the inlet. When the first particles reached outlet from the domain, the averaging procedure was turned on. Then the simulation ran for another 1.5 s. The initial configuration of the simulation could be seen in the **Fig. 3**.

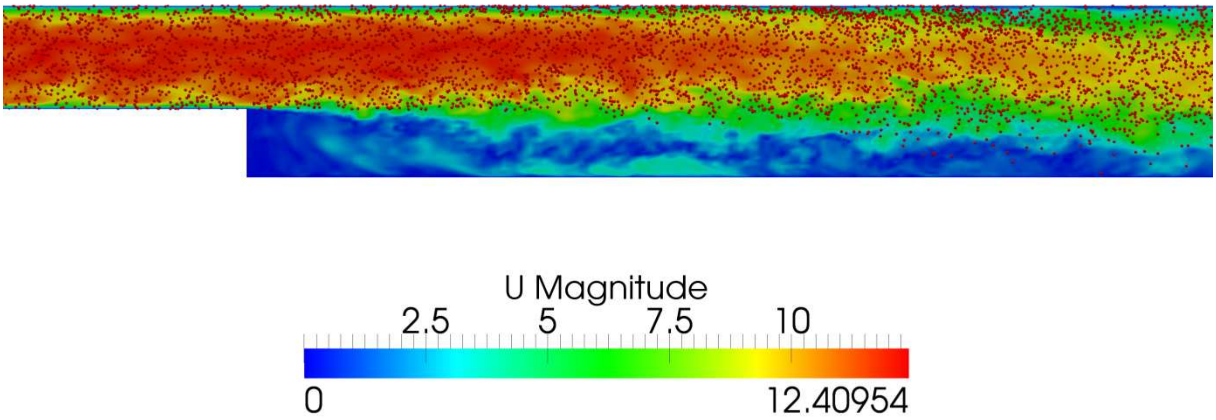

Fig. 3 Initial state of simulation

4 Results

The results of the simulations are in the **Fig. 4**. The streamwise velocity component along the height of the channel at a distance x / H = 7 is shown in this figure. The results of the experiment are in the **Fig. 5** for comparison. It is obvious that the particles tend to dampen the turbulence of carrier phase. It is caused by fact that the simulated particles are relatively small and light and are easily influenced by surrounding flow and they are often captured by vortices. The particles drain the turbulent kinetic energy from the vortices for their own motion. The consequence of these facts is decrease of the turbulent kinetic energy of the flow. The decrease of the turbulent kinetic energy occurs only in the upper part of the domain. The flow is not affected by particles in the lower region because in there are almost no particles in the wake at this distance from the step. Qualitatively, the simulation result matches the experiment. However, it appears that the degree of suppression of turbulence particles is underestimated.

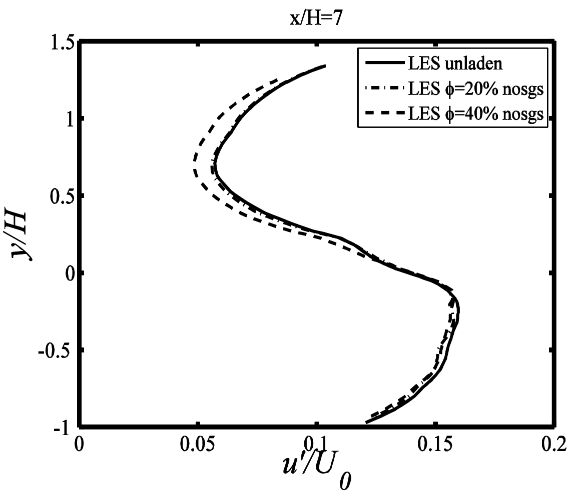

Fig. 4 LES simulation with particles

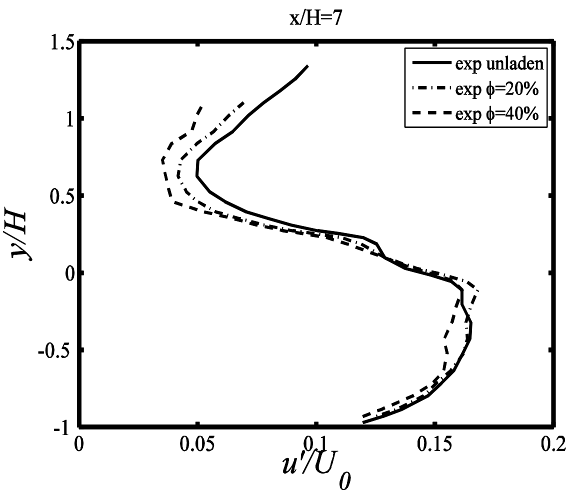

Fig. 5 Results of experiment

5 Conclusion

The simulations of the particle-laden backward facing step flow were performed. For the description of the system fluid-particle was used Euler-Lagrange approach. The simulation was done using two-way coupling. This means, that the influence of particles on the fluid was not neglected but was taken into account during simulation. The coupling between liquid and dispersed phase was achieved by adding special term to the momentum equation of motion. This term equals to the sum of forces acting on every particle. The concentration of the particles was chosen high enough to observe turbulence modulation. For the solution of the fluid motion was used Large Eddy Simulation method. The motion of the particles was described by Lagrangian equations of motion. These equations were solved using second order Euler scheme. The simulations were done for three different concentrations of particles in the region. It has shown that approach used in this work tends to underpredict the turbulence attenuation by particles. Another simulation with another types of particles will be done in order to get better understanding of the turbulence modulation by the particles. It will be also implemented modification proposed by [2], which includes the effect of the particles into subgrid model.

Acknowledgement The support of grants GA CR 101/08/0096 as well as the project FSI-S-11-6 is gratefully acknowledged.

Literatura

[1] FESSLER, J. R. & EATON, J. K. Turbulence modification by particles in a backward-facing step flow. J. Fluid Mech. 394, 1999, pp. 97-117

[2] GARCA, J. Study of the turbulence modulation in particle-laden flow using LES, AnnualResearch briefs, Centre for turbulence Research, 2001

[3] SOMMERFELD, M. & WACHEN, van B. & OLIEMANS, R. *Best Particle Guidelines*, ERCOFTAC, version 20-08-2007, 2007

[4] VOLAVY, J. & FORMAN, M. & JICHA, M. *Turbulence forcing scheme in physical space based on Ornstein-Uhlenbeck process*, CFD 2010, Lisbon, ISBN 978-889-96778-0-7, ECCOMAS, 2010