TURBULENCE MODULATION IN LARGE EDDY SIMULATION OF BACKWARD-FACING STEP FLOW LADEN WITH PARTICLES

Jaroslav Volavý*, Miroslav Jícha*

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. 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.

Keywords : *turbulence, Large Eddy Simulation, flow, particles*

1. Introduction

In many industrial areas we can encounter the processes in which the particles transportation by turbulent flow is involved, e.g. in the chemical or food industry, coal-fired power plants in the transportation of coal dust to the boiler, in the prediction of air pollution and many others. 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.

There were done many studies investigating the effect of the particles on the turbulence of the carrier phase in the past. The overview of these works is given by Gore and Crowe [1]. They have done summary of experimental data concerning systems gas/solid, gas/fluid, fluid/solid and fluid/gas phase in channels and jets. They identified the ratio of particle diameter and turbulence length scale to be suitable parameter for decision whether the turbulence will be enhanced or attenuated. Whenever is this ratio lower than 0.1 the turbulence is attenuated; for higher ratio the turbulence is enhanced. However, this work does not deal with quantitative changes of turbulence, only provides the set of parameters on which the change of intensity could be dependant. Another study was done by Kulick et al. [2]. The aim of their study was to estimate the quantitative changes of the turbulence

* Ing. J. Volavý, Ph.D., prof. Ing. M. Jícha, CSc., Brno University of Technology, Faculty of Mechanical Engineering, Energy Institute, Technická 2896/2, Brno

intensity with particles in the stream. They performed series of measurement in the vertical fully developed turbulent channel flow laden with particles. Measurements were done for different particle types and for various mass fractions of particles in the flow up to 0.8. They determined that the attenuation of the turbulence raise with increasing Stokes number of the particles. They also showed that the turbulence is more attenuated in the region with high spatial frequencies. Fessler and Eaton [3] experimentally studied turbulence modulation by particles on more complicated case of turbulent flow of vertical backward-facing step flow. They used glass particles with diameter of $90\,\mu$m and $150\,\mu$m and copper particles with diameter of $70\,\mu$m. The mass loading of particles in the flow varies from $20\,\%$ to $60\,\%$. They identified that the level of the turbulence attenuation increases with increasing mass fraction of the particles.

The numerical studies of the particle-laden flows were done also. Squires and Eaton [4] performed DNS simulation of forced homogeneous turbulent flow with particles. They have showed that heavier particles which react worse on the changes in the flow are more uniformly distributed in space and these particles cause more homogeneous modifications in the turbulence than the light particles. Light particles tend to cluster in the regions with high shear stress and low vorticity. They have also discovered that the additional source term from particles in the equation of motion acts as a dump for turbulent kinetic energy. The turbulence dissipation increased because of presence of small particles. The simulation of free decaying isotropic turbulence was done by Elghobashi and Truesdell [5]. Particles with relaxation time almost identical with Kolmogorov time scale were used in their study. They concluded that the turbulence decays faster with the particles than without them.

Yamamoto et al. [7] performed very detailed study of vertical particle-laden channel flow based on the experiments done by Kulick et al [2]. They used Large Eddy Simulation (LES) method for the solution of the liquid phase and Lagrangian approach for particles. They considered not only two-way interactions of particle-fluid, but they included the inter-particle collisions in the simulations as well. They showed that the inclusion of the inter-particle collisions (so called four-way coupling) plays important role in the prediction of the turbulence statistics in the flows with high particle concentrations. Another Large-eddy simulation of a vertical turbulent channel flow laden with a very large number of solid particles was performed by Vreman et al. [8]. The particle volume fraction equals about $1.3\,\%$, which is relatively high in the context of modern LES of two-phase flows. The method incorporated four-way coupling, i.e., both the particle-fluid and particleůparticle interactions were taken into account. The results showed that due to particle fluid interactions the mean fluid profile is flattened and the boundary layer is thinner. The comparison of two-way and four-way coupling clearly demonstrates that the collisions have a large influence on the main statistics of both phases. Similar study as Yamamoto et al. [7] has done Garcia [9]. He neglected the inter-particle collisions. The effect of the particles on the fluid is not included only in Navier-Stokes equations in form of coupling term, but the influence of the particles is included in subgrid model.

In this work we studied the influence of particles 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 effect of the inter-particle collisions was not considered due the low concentrations of the particles. The backward-facing step flow laden with particles was simulated. The geometric configuration of the backward-facing step, the flow parameters and particle properties used for simulations were chosen to be identical with experiment done by Fessler and Eaton [3]. The results of the simulations are compared with the results of the experiments. The simulations were carried out in the OpenFOAM, which is freely available and its source code is accessible.

2. Mathematical model

The system of carrier phase (liquid) and dispersed phase (particles) is described in this work using Euler-Lagrange approach. This means that the liquid is considered to be continuum and its motion is described by the Euler equation of motion. The particles are considered to be mass points and for their simulation is used Lagrangian approach. For each particle is formed equation of motion based on Newton's second law.

2.1. Liquid phase

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 [10] :

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

where $\bar{u}_i$ is the i-th component of the filtered velocity, $\bar{p}$ is filtered pressure and τ_{ij} is subgrid stress tensor defined as $\tau_{ij} = \overline{u_i\,u_j} - \bar{u}_i\,\bar{u}_j$. The standard Smagorinsky model [11] was used for evaluation of the subgrid stress tensor :

$$\begin{gathered} \tau_{ij} - \frac{1}{3}\,\delta_{ij}\,\tau_{kk} = -2\,\nu_\mathrm{t}\,\bar{S}_{ij}\ , \qquad \nu_\mathrm{t} = (C_\mathrm{S}\,\Delta)^2\,|\bar{S}|\ , \\ \bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\ , \end{gathered} \tag{2}$$

where $|\bar{S}| = |2\,\bar{S}_{ij}\,\bar{S}_{ij}|^{1/2}$, δ_{ij} is Kronecker delta and C_S is Smagorinsky constant. The value of this constant was equal to 0.17.

The influence of the particles on the fluid is obtained in the mathematical model using the action-reaction principle. The negative force acting on particles that is caused by surrounding fluid is added to the Navier-Stokes equations. This is represented by the coupling term f_i in the equation (1). This term represents the sum of forces acting on all particles. This term is defined as:

$$f_i(\mathbf{x}) = \sum_{n=1}^{N} \delta_{\mathbf{x},\mathbf{x}_n}\,f_{n,i}\ . \tag{3}$$

$f_{n,i}$ is i-th component of the force acting on the n-th particle, $\delta_{\mathbf{x},\mathbf{x}_n}$ is Dirac delta ($\delta_{\mathbf{x},\mathbf{x}_n} = \delta(\mathbf{x} - \mathbf{x}_n)$; $\mathbf{x}_n$ is position of n-th particle). Force $f_{n,i}$ is equal to the right hand side of equation (4). The force field f_i has non-zero value only in the positions of particles.

2.2. Solid phase

The motion of the particles is described by Lagrangian equations of motion. The particles are considered to be dimensionless mass points. The trajectories and the velocity of the particles are obtained by solving the particles equations of motion. These equations are based on the Newton's second law. There is broad spectrum of forces which can act on particles. In our study we considered only drag force. Another force which could be taken into consideration is gravity force. But due the geometry configuration when the flow is directed downwards and the gravity causes no acceleration the gravity plays no important role and thus it can be neglected. The experiment which the simulations are based on is designed in the way to eliminate the effect of gravitation. Another forces (like Basset force, Saffman force etc. [12]) can be neglected due the small size of particles. The equation of motion for j-th particle has form:

$$\frac{\mathrm{d}\mathbf{v}_j}{\mathrm{d}t} = \frac{\mathbf{u}(\mathbf{x}_j, t) - \mathbf{v}_j}{\tau_\mathrm{p}} \left(1 + 0.15\,\mathrm{Re}_\mathrm{p}^{0.687}\right) . \tag{4}$$

$\mathbf{v}_j$ is the particle velocity, $\mathbf{u}(\mathbf{x}_j, t)$ is velocity of the fluid seen by particle in position $\mathbf{x}_j$ and $\tau_\mathrm{p} = \varrho_\mathrm{p}\, d_\mathrm{p}^2/(18\,\varrho_\mathrm{f}\,\nu)$ is relaxation time of particle. The particle relaxation time defines how the particle reacts on the changes in the flow. The small light particles with low relaxation time tend to follow the flow more closely. They are also easily captured by vortices. On the other hand, the large heavy particles are not so sensitive to the fluctuations.

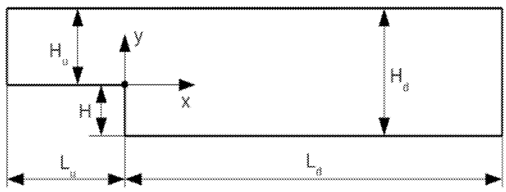
The linear interpolation was used for the interpolation of the velocity from Eulerian grid on which is fluid motion solved to the positions of each individual particle. The last term of equation (4) in the brackets represent the standard drag correlation for particles with particle Reynolds number greater than one.

3. Backward-facing step

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. This type of flow contains massive separation on the trailing edge and consequent re-attachment behind the step. Separation bubble on the wall opposite to the step could appear for some geometrical and flow parameters. The turbulent backward-facing step is very good benchmark for the various turbulent model. Good turbulent model should be capable of precise prediction of the distance behind the step of the re-attachment point.

The geometric configuration, flow parameters and boundary conditions are based on the experiment described in [3]. The channel is positioned vertically, the acceleration of gravity acts in the direction of flow. Inlet channel height is 40 mm and abruptly expands to 66.7 mm. Hence expansion factor is 5/3.

The ratio of the width of the domain in 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. For the simulation, however, this option represents a complication of a rise in the number of grid cells, thereby increasing computing demands. Therefore, it was only simulated 120 mm width slot and periodic boundary condition was prescribed on the sides. This choice eliminates the effect of the side walls.



channel height H_u	40 mm
channel length L_u	120 mm (3*H_u)
step height H	26.7 mm
H_d	66.7 mm
L_d	540 mm (14*H)
Expansion ratio	5:3 (H_d:H)

Tab.1: Geometry parameters of the backward-facing step

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 maximum size of the computational cell in the streamwise and spanwise direction is $\Delta x^+ \approx 28$ and $\Delta z^+ \approx 13$. The detail of the mesh near the trailing edge is in the Fig. 1.

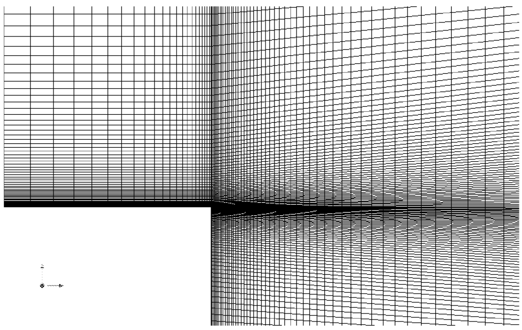


Fig.1: Detail of computational mesh

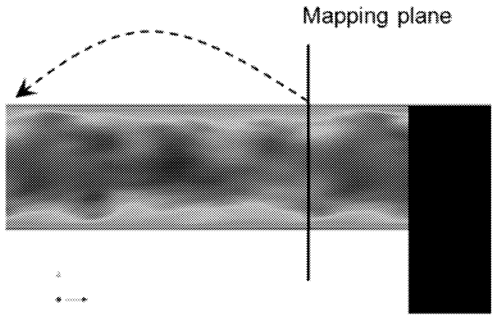


Fig.2: Scheme of inlet data mapping

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. Forcing scheme based on Ornstein-Uhlenbeck process was used [13] in order to accelerate the transition to turbulence. 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 plane. 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. This rather complicated way of inlet velocity treatment showed to be necessary in order to predict the flow accurately. Simple imposition of the uniform velocity profile leads to the occurrence of the separation bubble on the wall opposite to the step. Even prescribing the turbulent velocity profile did not eliminate this effect. More detailed information about the influence of the turbulence representation at the inlet plane and about single phase flow can be found in previous work of authors [16].

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, which suppose that the particles are regarded as mass point. Whenever the particles are bigger than the computational cell, it is not possible to use this approach. Another method must be applied, for example immersed boundary method [14].

The mass loading of the particles in the flow is 20 % and 40 %. These mass loadings lead to volume fractions of particles of 8×10^{-5} and 1.6×10^{-4}. According to [12], particles with these concentrations should affect the fluid. The turbulence modulation should be noticeable for volume fraction of particles about 8×10^{-5}.

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 which is approximately equal to the time of 30 passages of the fluid through the domain. The velocity data was sampled and used for calculation of average and fluctuating velocity filed over this time interval. This time interval has shown to be long enough for stabilizing of turbulent statistics. The average value and variance of the velocity in various points in the domain were observed during simulation and after this time these statistics showed no further changes. The initial configuration of the simulation could be seen in the Fig. 3.

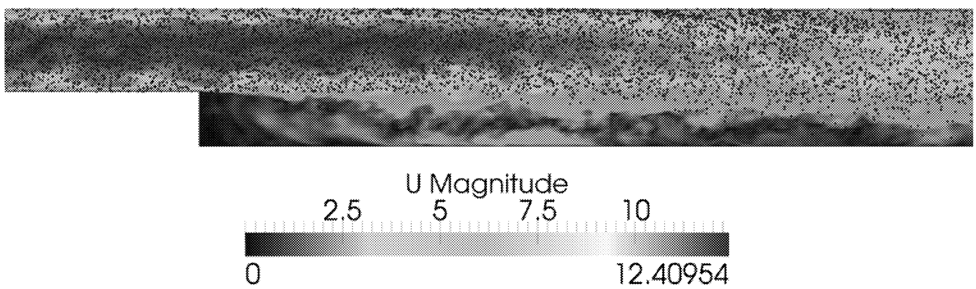


Fig.3: Initial state of simulation

The central difference scheme of the second order of accuracy was used for the discretization of the convective terms. The Crank-Nicolson scheme of second order of accuracy was applied for the time disretization. The Navier-Stokes equations were solved using PISO algorithm.

4. Results

Results of the simulations are described in this section. Results of the unladen flow are introduced first, and then follow results for particle laden flows with different mass fractions particles. The simulations were performed in open-source CFD code OpenFOAM 1.7-x.

In the figure Fig. 4 are depicted mean streamwise velocity profiles in distance $x/H = 7$ from the step. The full line represents LES simulation without particles; circles refer to the experimental data taken from [3]. From these pictures could be noticed, that the core of the stream is not so inclined downwards as the experiment. This resulted in overprediction of the distance of the reattachment point behind the step. The reattachment point predicted by simulation is located at position $x/H = 8.2$, experiment predicts this point at position $x/H = 7.4$. Better prediction of the reattachment point could be achieved by using more advanced subgrid model, for example localised Smagorinsky model proposed Ghosal et al. [15].

The mean streamwise velocity fluctuations are shown in the figure Fig. 5. Here can also be seen the lower inclination of the flow, the peaks are little bit shifted higher. However the difference between simulation and experiment is not crucial and the LES simulation could be considered sufficiently accurate.

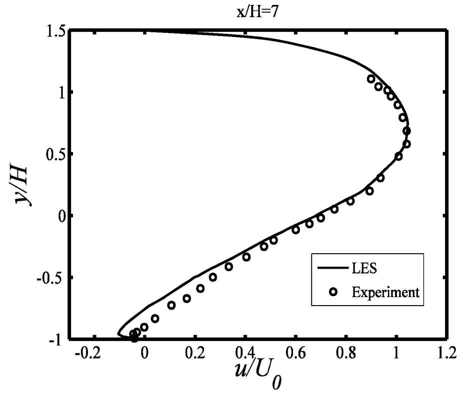


Fig.4: Mean streamwise gas-phase velocity

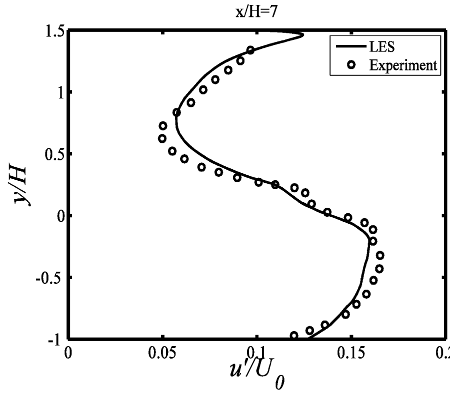


Fig.5: Steamwise velocity fluctuations

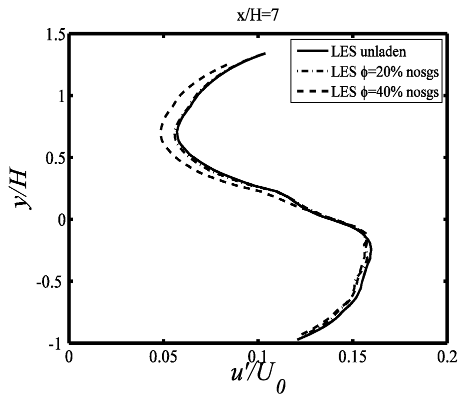


Fig.6: LES simulation with particles

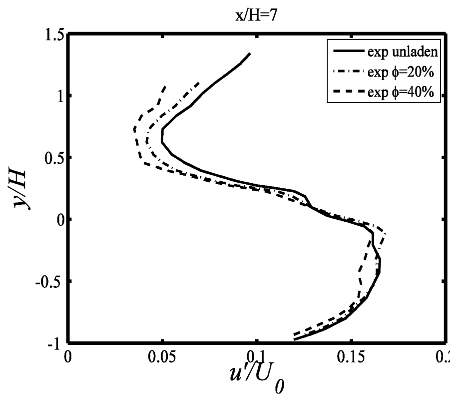


Fig.7: Results of experiment

The results of the simulations with particles are in the Fig. 6. The fluctuation of 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. 7 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 by particles is underestimated.

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 Garcia [9], which includes the effect of the particles into subgrid model.

Acknowledgement

The support of Netme centre – CZ.1.05/2.1.00/01.0002 as well as the project FSI-S-11-6 is gratefully acknowledged.

References

[1] Gore R.A., Crowe C.T.: Modulation of Turbulence by a Dispersed Phase, J. Fluid Eng. 113, 1991

[2] Kulick J.D., Fessler J.R., Eaton J.K.: Particle response and turbulence modification in fully developed channel flow, J. Fluid Mech. 277, 1994

[3] Fessler J.R., Eaton J.K.: Turbulence modification by particles in a backward-facing step flow, J. Fluid Mech. 394, 1999, pp. 97–117

[4] Squires K.D., Eaton J.K.: Particle response and turbulence modification in isotropic turbulence, Phys. Fluids A, Vol. 2, No. 7, 1990

[5] Elghobashi S., Truesdell G.C.: On the two-way interaction between homogeneous turbulence and dispersed solid particles. I: Turbulence modification, Phys. Fluids A, Vol. 5, No. 7, 1993

[6] Boivin M., Simonin O., Squires K.D.: On the prediction of gas-solid flows with two-way coupling using large eddy simulation, Phys. Fluids, Vol. 12, No. 8, 2000

[7] Yamamoto Y., Pottho M., Tanaka T., Kajishima T., Tsuji Y.: Large Eddy Simulation of turbulent gas-particle flow in a vertical channel: effect of considering inter-particle collisions, J. Fluid Mech., Vol. 442, 2001

[8] Vreman B., Geurts B.J., Deen N.G., Kuipers J.A.M., Kuerten J.G.M.: Two- and Four-Way Coupled EulerůLagrangian Large-Eddy Simulation of Turbulent Particle-Laden Channel Flow, Flow Turbulence Combust, 82, 2009

[9] Garcia J.: Study of the turbulence modulation in particle-laden flow using LES, Annual Research briefs, Centre for turbulence Research, 2001

[10] Sagaut P., Deck S., Terracol M.: Multiscale and Multiresolution Aproaches in Turbulence, Imperial College Press, London, ISBN 1-86094-650-X, 2006

[11] Smagorinsky J.: General circulation experiments with the primitive equations, Mon. Weather Rev., Vol. 91, No. 3, pp. 99–164, 1963

[12] Sommerfeld M., Van Wachen B., Oliemans R.: Best Particle Guidelines, ERCOFTAC, version 20-08-2007, 2007

[13] 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
[14] Peskin C.S.: The fluid dynamics of heart valves: Experimental, theoretical, and computational methods, Annu. Rev. Fluid Mech. 14, 235 (1982)
[15] Ghosal S., Lund T.S., Moin P., Akselvoll K.: A dynamic localization model for large-eddy simulation of turbulent flws, J. Fluid Mech. 286, 1995
[16] Volavý J., Forman M., Jícha M.: Influence of the turbulence representation at the inlet on the downstream flow pattern in LES of backward-facing step, Experimental fluid mechanics 2011: Conference proceedings, pp. 485–489, ISBN 978-80-7372-784-0, Technical university of Liberec, 2011

Received in editor's office: August 31, 2012
Approved for publishing: May 20, 2013