NEW INLET BOUNDARY CONDITION TREATMENT FOR LARGE EDDY SIMULATION

Jaroslav Volavy / Matej Forman / Miroslav Jicha
Faculty of Mechanical Engineering, Brno University of Technology, Brno, Czech Republic

Abstract
This work describes new treatment of inlet boundary condition for Large Eddy Simulation. The turbulence on the inlet is forced by generation of the force in particular points of the computational domain and this force is added to the momentum equations as a source term. For the generation of the force is used Ornstein-Uhlenbeck process. Ornstein-Uhlenbeck process describes the distribution of velocity fluctuations in homogeneous turbulence flow. Instead of generating random velocity fluctuations which would lead to continuity error the Ornstein-Uhlenbeck process is used here for generating the forcing force. This approach allows easy implementation in solvers of turbulent flows. This proposed inlet boundary condition was implemented in Large Eddy Simulation solver in open-source program OpenFOAM and then used for simulation of channel flow. The results are validated against DNS data. The results obtained with proposed forcing show good agreement with DNS results. The part of this work is also parameter study of influence of each parameter on witch depend Ornstein-Uhlenbeck process and their impact on the resulting turbulence field.

Introduction
In the simulation of turbulent flow is the most important issue proper choice of turbulence model that will provide good representation of the flow. Another issue of great importance is specification of boundary conditions, especially inlet boundary condition. The velocity and another inflow data linked with turbulence prescribed by boundary condition should be consistent with chosen turbulent model.

For the RANS simulations is sufficient to describe mean velocity profile and other turbulence variable (turbulent kinetic energy, ...) obtained from analytical solution or from experiments. This approach was justified in [7]. It was shown, that RANS model reach universal asymptotic behavior irrespective of the initial conditions. For Large Eddy Simulation, where the field on the inlet is turbulent, is situation more problematic. Usually the description of the flow is limited by knowledge of some statistical quantities such as mean velocity profiles and mass fluxes. In LES, the data generated by inlet boundary condition should consist of an unsteady turbulent velocity signal representing turbulence at the inlet. Ideally the simulation of upstream flow entering the computational domain will give good representation of the flow. However indefinitely extension in upstream direction is not possible because high computational cost. Therefore approximate inlet conditions must be specified.

Many methods were developed in the past. Kaltenbach et al. [8] used recycling method. Turbulent fluctuations are specified by running precursor simulation, whose only role is to provide main simulation with accurate boundary data. Another approach is to add synthetic turbulence at the inlet. Lund et al. [9] modified inlet velocity field by adding a random term to all velocity components. Klein et al. [10] proposed digital signal processing procedure to remedy lack of large-scale dominance in the inflow data generated by the random method.

Inlet data generation scheme

The scheme proposed in this work providing synthetic turbulence at inlet is applied in physical space. At the inlet boundary condition are defined several forcing points. In these points is consequently generated forcing force. This generation is done in every time step. This force is than added to the momentum equations of motion. The force is obtained by realization of Ornstein-Uhlenbeck process. The Ornstein-Uhlenbeck process is stochastic diffusive process generated by Langevin equation which gives a reasonable approximation for modeling of velocity fluctuations [4]. The force generated in one point is independent on the forces generated in other points. Therefore the number of realizations of Ornstein-Uhlenbeck process is equal to the number of defined points.

The generation of the forcing force is governed by equation (1):

$$F_i(t+\Delta t) = F_i(t) - F_i(t)\frac{\Delta t}{T_{OU}} + \left(\frac{2\sigma_{OU}^2 \Delta t}{T_{OU}}\right)^{1/2} \xi(t), \tag{1}$$

where Δt is time step of the simulation, T_{OU} and σ_{OU} are two input parameters of the process. Parameter T_{OU} characterizes the integral time scale of the process, σ_{OU} defines the variance of the process. Function $\xi(t)$ is the random variable with normal Gauss distribution (zero mean, unit variance). The individual points are distinguished by the subscript of i.

Here proposed forcing scheme has two input parameters: T_{OU} and σ_{OU}. There arises a question, how to set these parameters. Our suggestion is setting the integral time scale of the process T_{OU} equal to the integral time scale of the flow and the variance σ_{OU} equal to the bulk velocity.

Governing equations

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 modelled using subgrid model. The scale separation is done by applying low-pass filter operator on Navier-Stokes equation. If we apply the filter operator on Navier-Stokes equations we obtain filtered Navier-Stokes equations:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\bar{u}_i \bar{u}_j) = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_k \partial x_k} - \frac{\partial \tau_{ij}}{\partial x_j} + F, \tag{2}$$

where F is force defined by (1). For evaluation of subgrid stress tensor τ_{ij} is used subgrid kinematic energy model:

$$\tau_{ij} = -2\nu_k \bar{S}_{ij} + \frac{2}{3} k_{sgs} \delta_{ij}, \tag{3}$$

where $\bar{S}_{ij} = \partial \bar{u}_i / \partial x_j + \partial \bar{u}_j / \partial x_i$ and k_{sgs} is obtained from equation:

$$\frac{\partial k_{sgs}}{\partial t} + \bar{u}_i \frac{\partial k_{sgs}}{\partial x_i} = -\tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - C_c \frac{k_{sgs}^{3/2}}{\Delta} + \frac{\partial}{\partial x_j}\left(\frac{\nu_k}{\sigma_k}\frac{\partial k_{sgs}}{\partial x_i}\right). \tag{4}$$

The constants in equation (4) are set as follows: $C_k = 0.05$, $C_c = 1.0$ a $\sigma_k = 1.0$.

Description of the test case

For validation of the results obtained by proposed turbulence forcing shceme was chosen the case of turbulent channel flow. The simulated flow field is a fully developed turbulent flow between two parallel walls. Hence, the flow is homogeneous both in the streamwise and spanwise directions and the statistics are dependent only upon the distance from the wall. DNS data for comparison was taken from DNS database of fully developed turbulent channel flow [2]. The particular configuration of the test case come from choice of channel half-width $\delta = 0.1m$ and kinematic viscosity $\nu = 10^{-5}m^2s^{-1}$.

The simulations were performed for for high Reynolds number ($Re_\tau = 400$). The dimensions of the channel are: $5\Pi\delta$ in streamwise direction, 2δ in wall-normal direction and $2\Pi\delta$ in spanwise direction. For given kinematic viscosity $\nu = 10^{-5}m^2s^{-1}$ is bulk Reynolds number $Re_m = 14,000$ for $Re_\tau = 400$. The computational grid consists of 60x50x50 cells. The spacing both in streamwise and spanwise direction is uniform. The mesh becomes finer towards the wall in order to capture turbulence generation in the near-wall region. On the wall is satisfied condition $y^+ = yu_\tau/\nu \approx 1$. The grid spacing is in the streamwise and spanwise direction $\Delta x^+ = 28$ and $\Delta z^+ = 13$. The geometry and the computational grid is on the figure 1.

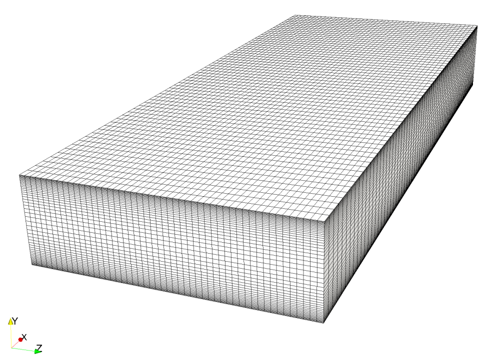

Figure 1: Geometry and mesh

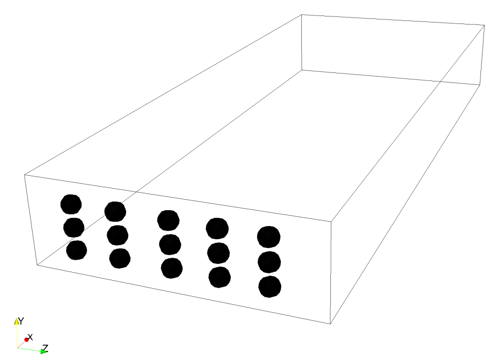

Figure 2: Location of forcing points

Boundary conditions in the area were set as follows: For $y = 0$ and $y = 2\delta$ is defined solid wall boundary condition. In the direction of the axis z (spanwise) is defined periodic boundary condition, in the direction of the x-axis (i.e. the entry and exit from the domain) is defined inlet and outlet boundary condition. Points where the force is generated are uniformly distributed over the inlet area to the domain, picture 2.

The proposed scheme was implemented into solver of turbulent channel flow *channelFoam* using Large Eddy Simulation method. The solver *channelFoam* is part of the open-source CFD software package OpenFOAM-1.6.x. The LES computations were performed using top-hat filter and subgrid kinematic energy model as a subgrid model.

Parametric study investigating the influence of number of forcing points on the flow was done. It has shown, that the increase number of forcing points brought no significant improvement.

Results

In the following paragraphs are given the results obtained with proposed turbulence forcing scheme. These results are represented by full line in the figures (labeled as Forcing), the dashed line refer to DNS data [2].

The velocity profile for high Reynolds number is shown in the picture 3. There is slightly underestimation of velocity in the near-wall region.

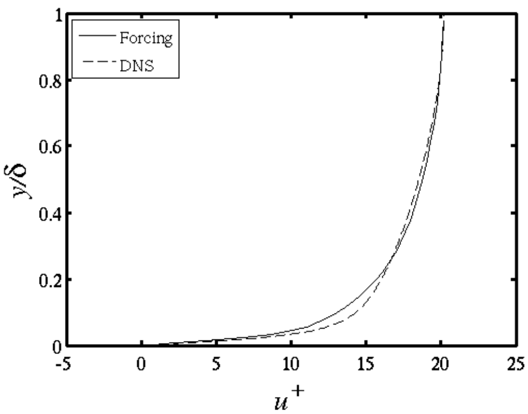

Figure 3: Velocity profile across the channel

The first moment statistics of high Reynold number channel flow are in pictures 4 and 5. The agreement between our LES and DNS is still good. The streamwise velocity fluctuation u^+_{rms} are underestimated near the wall, for $y/\delta > 0.3$ is this statistic overestimated. The opposite situation holds for wall-normal velocity fluctuations v^+_{rms}. In the near-wall region are these fluctuations overestimated, in the rest of the channel are underestimated. The peak of the v^+_{rms} is also flatter and further from wall than the DNS.

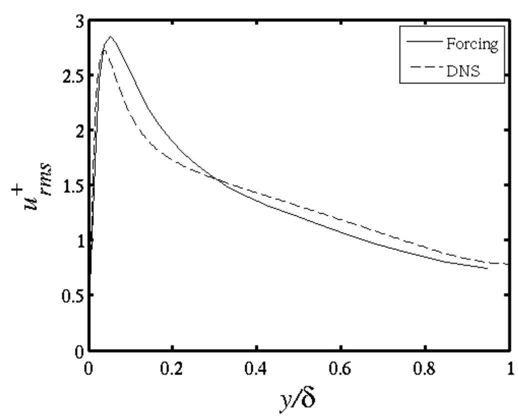

Figure 4: Streamwise velocity fluctuation

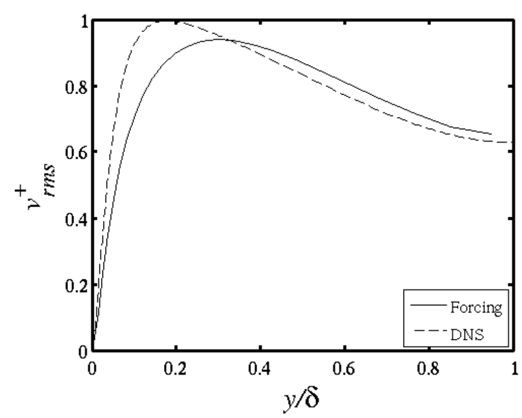

Figure 5: Wall-normal velocity fluctuation

The power spektra at different distances from the wall of high Reynolds flow are on figures 6 and 7. Agreement between LES with forcing and DNS is good. The value of the power spektrum for high vawenumbers are a bit underestimated by the LES.

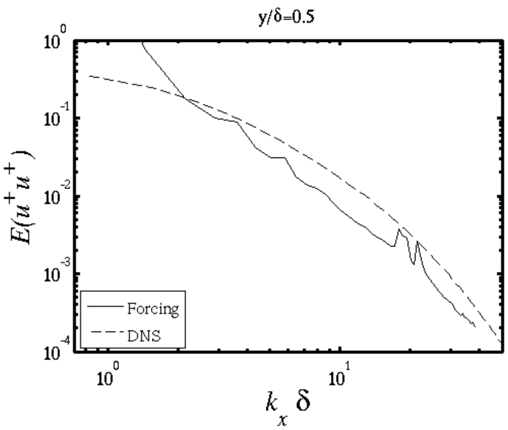

Figure 6: Energy spectrum in $y/\delta = 0.5$

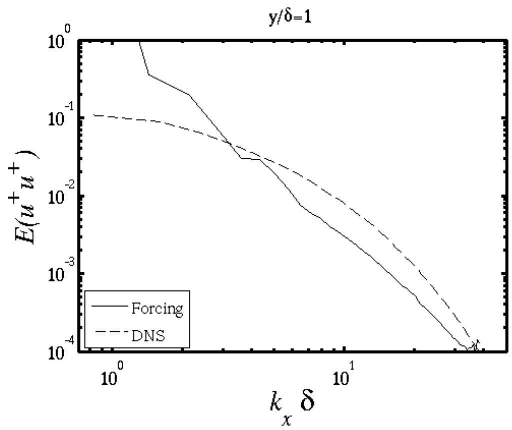

Figure 7: Energy spectrum in $y/\delta = 1$

Conclusions

New scheme of generating inflow velocity for LES was introduced in this work. This scheme operates in physical space. The main idea of this scheme is generating a force in every time step in particular points in the computational domain and this force is than added to momentum equations. This force is generated using Ornstein-Uhlenbeck stochastic process. The main feature of this process is that the force generated in next time step is not independent on the previous value (the Ornstein-Uhlenbeck is so called Markov process). So there is a correlation in the history of the generated force and it is suitable for generation of turbulence.

The developed forcing scheme was tested on the case of turbulent channel flow. Method for solution was Large Eddy Simulation with subgrid kinematic energy subgrid model. The results presented in this work are for high Reynolds number channel flow. In this case are the results in a good agreement with DNS data.

The effect of number of forcing points was examined too. It turned out that the increase of these point has no significant effect on the turbulence. It was too examined the case when the forces generated in forcing points has only wall-normal and spanwise component (the streamwise component is zero). In this case the results were almost identical therefore they were not introduced here.

Acknowledgment

The support of grant GA ČR 101/08/0096 as well as the project FSI-J-10-7 and COST Action P20 OC166 is gratefully acknowledged.

References

[1] C. Rosales and C. Meneveau, Linear forcing in numerical simulations of isotropic turbulence: Physical space implementations and convergence properties, Phys Fluids 17 (2005)

[2] K. Iwamoto, Y. Suzuki and N. Kasagi, Database of fully developed channel flow, Internal report (http://www.thtlab.t.u-tokyo.ac.jp/DNS/ILR-0201.pdf), University of Tokyo, Japan (2002)

[3] M. Overholt and S.B. Pope, A deterministic forcing scheme for direct numerical simulations of turbulence, Comput. Fluids, vol. 27, no. 1, (1998) 11-22

[4] S.B. Pope, Turbulent Flows, Cambridge University Press, Cambridge, ISBN 0-521-59125-2 (2000)

[5] T.S. Lundgren, Lineary forced isotropic turbulence, Annual Research Briefs (Center for Turbulence Research), Stanford (2003) 461-473

[6] V. Eswaren and S.B. Pope, An examination of forcing in direct numerical simulations of turbulence, Comput. Fluids, vol. 16, no. 3, (1988) 257-278

[7] W.K. George and L. Davidson. Role of initial conditions in establishing asymptotic flow behavior. AIAA Journal, 42(3), (2004) 438446

[8] H.J Kaltenbach, M. Fatica, R. Mittal, T.S. Lund, and P. Moin. Study of flow in a planar asymmetric diffuser using large eddy simulation. Journal of Fluid Mechanics, (1999) 151185

[9] T. Lund, X. Wu, and D. Squires. Generation of turbulent inflow data for spatially-developing boundary layer simulations. Journal of Computational Physics, (1998) 233258

[10] M. Klein, A. Sadiki, and J. Janicka. A digital filter based generation of inflow data for spatially developing direct numerical or large-eddy simulations. Journal of Computational Physics, 186, (2003) 652665