Numerical modelling of wall-bound turbulent flows based on the CABARET scheme

Danil Asfandiyarov
Nuclear safety institute of Russian academy of sciences, 52, Bolshaya Tulskaya Street, Moscow, 115191, Russia
E-mail: dasfandiyarov@ibrae.ac.ru

Abstract. In this study, the classical problem of turbulent plane channel flow at \( Re_m = 13750 \) and \( Re_m = 21900 \) is investigated using a numerical algorithm based on the CABARET scheme. Simulations are performed on the fine grids, resolving all the spatial scales of the turbulence according to a direct numerical simulation approach, as well as on coarse grids standard for large eddy simulation of wall-bounded turbulent flows. In the latter case, in order to obtain a more accurate representation of the momentum transfer towards the wall, artificial boundary conditions are introduced. It allows modelling with high accuracy the mean flow characteristics. Numerical results obtained by the CABARET scheme are compared with numerical results obtained by pseudospectral method.

1. Introduction
At the present time, modelling of turbulent flows is a fundamental problem, which, apart from its scientific importance, has been at the core of numerous technical applications. The most complete description of turbulence can be obtained within direct numerical simulation (DNS) where all spatial scales of turbulence are resolved. At the present stage, DNS approach is constrained by computational performance of modern supercomputers and numerical algorithms. Therefore, considerable efforts have been made to develop and verify various scale-resolving methods that would provide a reasonable balance between the depth of physical description and computational resources needed to solve the Navier-Stokes equations.

A classical large eddy simulation (LES) approach is based on separation of the large-scale motions from the small-scale ones, which are closer to isotropic. The large-scale components of flow variables are explicitly integrated in time, and interactions with the unresolved small-scale components are modelled. LES demonstrated for a wide range of problems that it calculates with high accuracy the mean characteristics of flow as well as the moments of a higher order. Near-wall turbulence poses the most difficult problem to the LES approach. The flow ceases being isotropic and homogeneous near the wall. In the vicinity of the solid boundary the scales of large and small eddies are overlapped and generation of the turbulent energy occurs at the scales comparable with the distance from the wall. As well as in a DNS approach LES requires very detailed structural information on the fluctuations to yield accurate results.

Development of novel efficient LES-algorithms that would allow for a correct representation of the complex dynamical behaviour in the near-wall layer is one of the main problems in LES modelling. Development of implicit LES methods (ILES) based on high-resolution numerical
schemes is at the cutting edge of LES modelling. In such schemes, the dissipative mechanism (smoothing filter) is provided by the operator approximating convective terms. A required amount of dissipation is introduced by a non-linear flux correction procedure.

The CABARET scheme [1], which belongs to this group of schemes, is distinguished by its simplicity, computational efficiency, improved dissipative and dispersive properties. The scheme is defined on a compact numerical template. It has a second order of approximation both in time and in space and allows the introduction of non-linear flux correction based on the maximum principle.

This paper presents the results of numerical simulation of incompressible turbulent channel flow based on the CABARET scheme for Reynolds numbers $Re_m = 13750$ and $Re_m = 21900$. Simulations are performed on both fine and coarse grids. In the latter case artificial boundary conditions are added. They can be viewed as an improvement of flow variables calculated in the grid cells in the vicinity to the wall and are introduced by using explicit subgrid-scale model accounting for shear effects in wall-bounded turbulent flows.

2. Governing equations
The Navier-Stokes equations for incompressible fluid were considered:

$$\frac{\partial u_i}{\partial x_i} = 0,$$

(1)

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 u_i.$$

(2)

Time integration of the set of equations (1)-(2) was provided by a computational method based on the CABARET scheme for orthogonal grids under the assumption of incompressible fluid [2]. Simulations on coarse grids were provided in the LES formulation with the closure in the form of an eddy viscosity model:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0,$$

(3)

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \nabla^2 \bar{u}_i + \frac{\partial (2\nu T S_{ij})}{\partial x_j},$$

(4)

where $S_{ij}$ is the filtered rate-of-strain tensor. In near-wall boundary cells the shear-improved Smagoronsky model [3] for the eddy viscosity $\nu_T$ was used:

$$\nu_T(x,t) = (C_s \Delta)^2 |S(x,t)| - ||S(x,t)||,$$

(5)

where $|S| = \sqrt{2S_{ij}S_{ij}}$. Here, the angle brackets $\langle ... \rangle$ denotes averaging over characteristic time intervals (in present work averaging is performed over 10000 time steps) [4]. The characteristic subgrid-scale $\Delta$ is determined from the volume of the cell $\Delta = (\Delta x \Delta y \Delta z)^{\frac{1}{3}}$. The Smagorinsky constant $C_s$ is assumed to be 0.16.

Shear-improved Smagorinsky model (5) allows for modelling of the energy scatter on the subgrid scale caused by both turbulent fluctuations and the mean gradient of velocity [3].

The components of the rate-of-strain tensor $S_{ij}$ and the eddy viscosity $\nu_T$ are calculated only in near-wall boundary cells (at the faces center). At the wall the value of the eddy viscosity $\nu_T$ is assumed to be zero. In other cells $\nu_T = 0$ and numerical dissipation works as a subgrid-scale model, due to the non-linear flux correction based on the maximum principle (ILES approach)[4].
3. Numerical simulations

The computational domain is a rectangular channel with two solid walls. Periodic boundary conditions were imposed in the streamwise and spanwise directions and no-slip conditions at the wall. At initial time, velocity distributions were designed to satisfy a Poiseuille profile plus a small random perturbation. The flow rate was kept constant. As soon as fully developed turbulent flow had been achieved, statistics were accumulated.

Non-uniform meshes, like in [4] and [5], with gradual concentration towards the walls and symmetrical relative to the central horizontal plane of the channel were used. In the periodic directions the meshes were uniform.

Fine meshes were used for the DNS simulations. The size of the boundary cells in the normal direction are to be $\Delta y^+ \approx 0.1 - 0.15$ in wall units. A LES approach mitigates the requirements to some extent with $\Delta y^+ \approx 1 - 1.5$. Here $y^+ = \frac{y u_\tau}{\nu}$, where $u_\tau$ is the friction velocity, $\nu$ is the kinematic viscosity.

The parameters of the calculation domain and the meshes are shown in the table 1. Here, the Reynolds number is defined as $Re_m = \frac{2 u_m \delta}{\nu}$, where $u_m$ is the averaged velocity, $\delta$ is the channel half width and $\Delta y^+_C$ is the size of the cells in the middle of the channel.

Table 1. Computational domain parameters.

| $Re_m$ | Dimensions | Mesh size | $\Delta x_+$ | $\delta z_+$ | $\delta y^+_W$ | $\Delta y^+_C$ |
|--------|-------------|-----------|--------------|--------------|----------------|----------------|
| 13750  | $2\pi \delta \times 2\delta \times \pi \delta$ | $512 \times 256 \times 256$ | 4.82         | 4.82         | 0.11           | 9.03           |
| 13750  | $4\pi \delta \times 2\delta \times 2\pi \delta$ | $160 \times 64 \times 160$ | 30.8         | 15.4         | 1.18           | 20.9           |
| 21900  | $2\pi \delta \times 2\delta \times \pi \delta$ | $512 \times 256 \times 512$ | 7.2          | 3.6          | 0.14           | 13.8           |
| 21900  | $4\pi \delta \times 2\delta \times 2\pi \delta$ | $256 \times 128 \times 256$ | 28.8         | 14.4         | 1.18           | 14.1           |

4. Numerical results

In figures 1-2 the mean velocity $u^+(y^+)$ is displayed as a function of the wall-normal distance $y^+$ for $Re_m = 13750$ and $Re_m = 21900$ respectively. The DNS and ILES results are represented at the same plot. The numerical results obtained by the CABARET scheme are compared with the DNS results obtained by the pseudospectral method [6].

The three normal components of the Reynolds stress tensor (normalized by the squared friction velocity) for both DNS and ILES are shown in figures 3-6.

The DNS results obtained by the CABARET scheme demonstrate a high degree of consistency with the results obtained by the pseudospectral method [6]. The average velocity profiles were calculated quite accurately on coarse meshes, while a slight deviation manifests in the profiles of normal stresses. The calculated values of the skin-friction coefficient are presented in the table 2.
Figure 1. The mean velocity profile normalized by the friction velocity in turbulent channel flow at $Re_m = 13750$. 1 - DNS (present), 2 - ILES (present), 3 - DNS of channel flow by pseudospectral method [6].

Figure 2. The mean velocity profile normalized by the friction velocity in turbulent channel flow at $Re_m = 21900$. 1 - DNS (present), 2 - ILES (present), 3 - DNS of channel flow by pseudospectral method [6].

Figure 3. Profiles of Reynolds stresses normalized by the friction velocity in turbulent channel flow at $Re_m = 13750$. DNS results. 1 - $u'^2$, 2 - $v'^2$, 3 - $w'^2$, 4 - DNS of channel flow by pseudospectral method [6].

Figure 4. Profiles of Reynolds stresses normalized by the friction velocity in turbulent channel flow at $Re_m = 21900$. DNS results. 1 - $u'^2$, 2 - $v'^2$, 3 - $w'^2$, 4 - DNS of channel flow by pseudospectral method [6].

5. Conclusions
In this study, the classical problem of viscous turbulent channel flow for various Reynolds numbers was investigated. The simulation results obtained by the CABARET scheme following the DNS approach demonstrate a good agreement with those calculated by the pseudospectral method, which is considered to be the most accurate for this type of flow. Moreover, ILES simulation results on relatively coarse meshes are given. The proposed computational algorithm with the use of artificial boundary conditions allows for accurate calculation of the first and second moments of the flow characteristics on such meshes.
Figure 5. Profiles of Reynolds stresses normalized by the friction velocity in turbulent channel flow at $Re_m = 13750$. ILES results. 1 - $u'^2$, 2 - $v'^2$, 3 - $w'^2$, 4 - DNS of channel flow by pseudospectral method [6].

Figure 6. Profiles of Reynolds stresses normalized by the friction velocity in turbulent channel flow at $Re_m = 21900$. ILES results. 1 - $u'^2$, 2 - $v'^2$, 3 - $w'^2$, 4 - DNS of channel flow by pseudospectral method [6].

Table 2. The skin-friction coefficient.

| $Re_m$ | $C_f$ | $C_f$, DNS Moser et al [6] |
|--------|-------|-----------------------------|
| 13750  | $6.51 \times 10^{-3}$ (DNS) | $6.50 \times 10^{-3}$ |
| 13750  | $6.50 \times 10^{-3}$ (ILES) | $6.50 \times 10^{-3}$ |
| 21900  | $5.75 \times 10^{-3}$ (DNS) | $5.75 \times 10^{-3}$ |
| 21900  | $5.78 \times 10^{-3}$ (ILES) | $5.75 \times 10^{-3}$ |

Acknowledgments
The research was carried out using the equipment of the shared research facilities of HPC computing resources at Lomonosov Moscow State University.

References
[1] Goloviznin V M, Zaitsev M A, Karabasov S A and Korotkin I A 2013 New algorithms of computational fluid dynamics for multiprocessor computing systems (Moscow: Moscow University Press)
[2] Asfandiyarov D G, Goloviznin V M and Finogenov S A 2015 Parameter-free method for computing the turbulent flow in a plane channel in a wide range of Reynolds numbers J. Comp. Math. Math. Phys.. 55 1515 26
[3] Leveque E, Toschi F, Shao L, Bertoglio J P 2007 Shear-improved Smagorinsky model for large-eddy simulation of wall-bounded turbulent flows J. Fluid Mech. 570 491 502
[4] Asfandiyarov D G 2019 J. Num. Meth. Prog. 20 12 20
[5] Asfandiyarov D G, Goloviznin V M and Finogenov S A 2015 J. Prob. Atom. Sci. Tech. Series: Math. Model. Phys. Proc. 2 48 58
[6] Moser R D, Kim J, Mansour N N 1999 Direct numerical simulation of turbulent channel flow up to $Re_\tau = 590$ Phys. Fluids. B 11 943 45