On mesh requirements for Large Eddy Simulation with Wall Functions

Dmitry K. Kolmogorov 1, Florian Menter2 and Andrey V. Garbaruk1

1 Peter the Great St. Petersburg Polytechnic University (SPbPU), Politechnicheskaya 29, 195251, Saint-Petersburg, Russia
2 ANSYS Germany GmbH

E-mail: dkolmogorov@cfd.spbstu.ru

Abstract. The results of using Large Eddy Simulation with Wall Functions (WFLES) in application to basic wall-bounded flows, such as turbulent boundary layer and channel flow cases are presented. In particular, it is shown that WFLES is suitable for predicting wall bounded flows and provides reasonable accuracy when using appropriate grids. The grid for WFLES should have isotropic cells with size smaller than 10% of the boundary layer thickness. Using coarser grids or anisotropic cells leads to significant reduction of accuracy.

1. Introduction

Nowadays approaches based on scale resolved simulations (SRS) are more and more often used for engineering computations of wall bounded flows. However the computational resources presently available are insufficient [1] for application of more accurate methods, such as Direct Numerical Simulations (DNS) and Large Eddy Simulation (LES) with resolution of boundary layer up to the wall (Wall Resolved LES - WRLES). The lack of computational resources stimulated the development of hybrid approaches, such as LES with near-wall RANS treatment (Wall Modelled LES - WMLES) [2]. Unfortunately the computational cost of these methods is very high as well. This fact stimulates attempts to develop less expensive methods.

One of the possible alternatives, ensuring satisfactory accuracy and moderate computational efforts is LES with Wall Functions (WFLES) [3,4]. This method can be used on meshes which do not resolve the internal part of the boundary layer, thus the size of the computational meshes can be decreased significantly. This makes the method very attractive for engineering computations of wall bounded flows. However, systematic investigations of the accuracy of this approach is not presented in literature and the main goal of the current work is defining mesh and scheme requirements for WFLES applied to wall bounded flows.

The paper is composed of three parts as follows. First, numerical schemes and turbulence model employed in computations are described. Second, problems statements are given. Then, results of WFLES computations and mesh recommendations are described. Finally, conclusions are given.

2. LES model and numerical methods

Computations within the paper are carried out with the use of the general purpose CFD code ANSYS Fluent [5], within which the governing equations for the incompressible fluid are solved.
in the transient formulation by means of the finite volume method on collocated grids. WALE turbulence model [6] with the coefficient $C_w = 0.5$ is used to model turbulent viscosity. The model is suitable for both free shear and wall bounded flows.

A modified version of the BCD scheme [7], namely the Weakly Bounded Central Difference (WBCD) scheme is used for discretisation of the convective terms. The scheme allows control of numerical dissipation using a weighting coefficient, with the optimal value for high quality meshes equal to 0.25. A 2nd order backward difference scheme is used for temporal discretization. The pressure gradient is discretised using a second order pressure scheme; gradient terms are approximated using the Least Square Cell Based (LSCB) scheme.

SIMPLEC [8] approach with 5 subiterations per time step is used to ensure residual drop up to 1 order for pressure and 4 orders for velocity components. Pressure velocity coupling is ensured using a modified version of Rhie-Chow interpolation [9]. A classical wall function treatment similar to Kader [10] is used.

3. Problem statement

Periodic Turbulent Channel flow is a basic test for SRS approaches to predict wall-bounded flows. Simulations of this flow have been carried out assuming incompressible fluid at several Reynolds numbers $Re_\tau = u_\tau \cdot h/\nu$ based on friction velocity $u_\tau$ and half of channel height $h$. The flow is driven by a constant pressure gradient $dp/dx = -\rho u_\tau/h$, where $p$ is the pressure and $\rho$ is the density. This pressure gradient is taken into account in the governing equations via a source term in the momentum equations, which allows imposing periodic boundary conditions not only in the spanwise direction $z$, but also in the streamwise direction $x$.

The size of the computational domain is $L_x = 12h$, $L_y = 2h$ and $L_z = 3h$, where $L_x$, $L_y$ and $L_z$ are the domain sizes in the vertical-, spanwise- and streamwise- direction, respectively. These dimensions are selected to provide enough space for the largest turbulence structures during the simulation. The time step $\Delta \tau$ is set such that the largest Courant number is 0.5. Standard Wall Functions [11] are used as boundary conditions at the top and bottom boundaries of the domain.

Periodic boundary conditions are set in streamwise and spanwise directions. For the initial conditions, a result of a precursor RANS computation with imposed fluctuations is used. The fluctuations were obtained with the use of the Volumetric Synthetic Turbulence Generator (VSTG) [12] method.

Turbulent boundary layer flow is another benchmark test case used for turbulence modeling. In the current work the Reynolds number based on momentum thickness and free-stream velocity is equal to $Re_\Theta = 10000$. A computational domain has the dimensions in $x$-, $y$- and $z$-directions $L_x/\delta = 30$, $L_y/\delta = 12$ and $L_z/\delta = 0.3$, respectively, where $\delta$ is the boundary layer thickness at the inlet. A velocity profile with imposed time-dependent fluctuations is set at the inlet boundary. Synthetic Turbulence Generator (STG) [12] is used at the inlet to generate turbulent structures. Periodic boundary conditions are set in the spanwise direction, and constant pressure is used at the outlet.

All simulations were performed in two stages. During the transient period a statistically mature solution was obtained. After that averaging was started. The transient period and period of averaging can be measured by means of the number of periods the flow passes the full domain. For channel flow the number is 50 for both transient and averaging steps. In the turbulent boundary layer case 8 flow passes are used for both transient and averaging steps.

4. Results

4.1. Results of WFLES

Turbulent channel flow at $Re = 18000$ and turbulent boundary layer flow at $Re_\Theta = 10000$ are considered on uniform cubic meshes with 20 cells per boundary layer (BL). As seen
from Figure (1(a)) mean velocity profiles in the turbulent channel flow case are in agreement with Reichardt’s law [13]. The mean shear stresses, however, have oscillations along vertical directions, see example on Figure (1(b)). In spite of the fact that using schemes with higher dissipation results in smoother mean fields, the oscillations were found to be nearly insensitive to the choice of numerical schemes, level of convergence, WALE model constant, type of wall function and can be probably explained by the high velocity gradient in the vicinity of the wall.

Similarly to the channel flow case the mean velocity profiles are well predicted in turbulent boundary layer case Figure (1(c)). $C_f$ distribution is in good agreement, see Figure (1(d)), with experimental data [14–16].

![Figure 1](image-url)

**Figure 1.** Results on uniform cubic meshes: a) and b) - channel flow; c) and d) - turbulent boundary layer flow.

### 4.2. Influence of cell size in WFLES

The minimum required number of cells per boundary layer is determined then for WFLES computations on uniform cubic meshes with 5, 10 and 20 cells per BL. As seen from Figure (2(a)) mean velocity profiles in the channel flow are in agreement with Reichardt’s law. Tests at other Reynolds numbers on meshes with 5 cells per BL show the good agreement with Reichardt’s law as well, see Figure (2(b)).

In the turbulent boundary layer case, the influence of cell size on the solution accuracy is stronger compared to channel flow, see Figure (2(c)), where the solution on all meshes, except for the mesh with 5 cells per BL, are in agreement with Reichardt’s law. $C_f$ distribution on mesh with 5 cells per BL is under-predicted, see Figure (2(d)). Results on the mesh with 10 cell per BL are in good agreement with reference data.
Figure 2. Sensitivity to cells size on uniform meshes in WFLES of channel flow a) and b), and turbulent boundary layer flow c) and d): a) mean velocity profiles at \(Re_\tau = 18000\); b) mean velocity profiles at various \(Re_\tau\) on mesh with 5 cells per boundary layer; c) mean velocity profiles at \(Re_\Theta = 10000\) in the turbulent boundary layer case and d) \(C_f\) distribution on uniform meshes.

4.3. Influence of cell aspect ratio

Uniform cubic meshes were used for the results described in the previous section. For practical applications meshes with cells with sizes in the \(x\), \(y\) and \(z\) directions, (respectively, \(\Delta_x\), \(\Delta_y\) and \(\Delta_z\)) that are not identical, are of primary interest. However, it is important to note that strong refinement of WFLES meshes in the wall normal direction alone would lead to a severe deterioration of the results.

Below the sensitivity of WFLES results to cell aspect ratio, \(K = \Delta_x/\Delta_y\), is tested in channel and turbulent boundary layer cases.

**Figure 3.** Example of cells with different various aspect ratio \(K\): 1, 2 and 4

For the channel flow case meshes with identical resolution along \(y\) direction are considered with \(\Delta_y = \delta/10\) and cells isotropic in \(x\) and \(z\) direction, i.e. \(\Delta_x = \Delta_z\), see examples of mesh cells on Figure (3). In the turbulent boundary layer case meshes with identical near wall resolution across the BL are considered such that near wall \(\Delta_y = \delta/10\).

As seen from Figure (4(d)) the distribution of friction coefficient in the turbulent boundary layer case is under predicted when cell aspect ratio \(K > 2\). For \(K = 4\) velocity profiles in the
channel flow case, see Figure (4(a)), and in the turbulent boundary layer case, see Figure (4(c)), are shifted up against Reichardt’s law. For clarity, the normal stresses and $C_f$ in the turbulent boundary layer case are compared with algebraic WMLES results obtained on a WMLES mesh [17].

Overall the results in both channel and the turbulent boundary layer cases are in good agreement with the reference on meshes with cell aspect ratio less or equal 2.

Figure 4. Influence on cells aspect ratio in channel flow case a), b) and the turbulent boundary layer case at section $x = 0.4$ c), d)

5. Conclusions
Results of WFLES applied to channel flow and the turbulent boundary layer problems show that WFLES ensures acceptable accuracy in basic wall bounded flow computations. It was shown that WFLES requires isotropic meshes with at least 10 cells per boundary layer in the wall-normal direction. It is recommended to use cubic meshes or at least grids with cell aspect ratio not larger than 2.

Acknowledgments
The results of the current work have been obtained with use of computational resources of Supercomputer Center at Peter The Great Saint-Petersburg Polytechnical University (www.spbstu.ru) and supported by the Academic Excellence Project 5-100 proposed by Peter the Great St. Petersburg Polytechnic University.

References
[1] Garbaruk A 2016 Peter the Great St.-Petersburg Polytechnic University
[2] Larsson J, Kawai S, Bodart J and Bermejo-Moreno I 2016 Mechanical Engineering Reviews 3 15-00418
[3] Launder B and Spalding D 1974 The numerical computation of turbulent flows
[4] Werner H and Wengle H 1993 Large-eddy simulation of turbulent flow over and around a cube in a plate channel Turbulent shear flows 8 (Springer) pp 155–168
[5] Mathur S and Murthy J 1997 Numerical heat transfer 31 195–215
[6] Nicoud F and Ducros F 1999 Flow, turbulence and Combustion 62 183–200
[7] Jasak H, Weller H and Gosman A 1999 International journal for numerical methods in fluids 31 431–449
[8] Van Doormaal J and Raithby G 1984 Numerical heat transfer 7 147–163
[9] Rhie C and Chow W 1983 AIAA journal 21 1525–1532
[10] Kader B 1981 International journal of heat and mass transfer 24 1541–1544
[11] Manual A F U 2016 ANSYS Inc
[12] Shur M L, Spalart P R, Strelets M K and Travin A K 2014 Flow, turbulence and combustion 93 63–92
[13] Reichardt H 1951 ZAMM-Journal of Applied Mathematics and Mechanics/Zeitschrift für Angewandte Mathematik und Mechanik 31 208–219
[14] Fernholz H and Finleyt P 1996 Progress in Aerospace Sciences 32 245–311
[15] Österlund J M 1999 Experimental studies of zero pressure-gradient turbulent boundary layer flow Ph.D. thesis Mekanik
[16] De Graaff D B and Eaton J K 2000 Journal of Fluid Mechanics 422 319–346
[17] Menter F R 2012 ANSYS Germany GmbH 1 17–18