Large Eddy Simulation with wall functions of Ahmed body

Dmitry K. Kolmogorov¹, Andreas Hüppe², 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. Large Eddy Simulation with Wall Function (WFLES) is known to be a cheap alternative to classical LES methods for simulation of flow where large and complex computational meshes are typically required. This makes it attractive for engineering applications. However experience of applying such methods to complex turbulent flows with flow separation and reattachment is still little-known in literature. In this work WFLES of flow around simplified car body with slant angle equal to 25 degrees and \( \text{Re}_L = 2.8 \cdot 10^6 \) is carried out on Octree mesh to demonstrate the capabilities and limitations of the method in such type of the flow. The results on a series of meshes show that even though the general flow topology is well captured, the critical part of the flow on the slant is hardly predicted even on 100 mln mesh. It is concluded that the prediction of separation above the slant requires significant mesh refinement even in the frame of WFLES.

1. Introduction

Nowadays capabilities of RANS models are known to be not sufficient for accurate prediction of turbulent flows with separation and reattachment. For improvement of the prediction accuracy of such flows the Scale-Resolving Simulation (SRS) methods are required, where at least a part of the flow is resolved, and another part is modeled. The most popular approaches are hybrid RANS-LES methods [1–3], which contrary to classical LES methods allow to significantly reduce the computational requirements due to near wall treatment using RANS models, however the methods still suffer from the limitations of the RANS model.

Contrary to hybrid methods classical LES approach combined with such models as Smagorinsky [4], WALE [5], Dynamical [6] and other suffer from near quadratic increase of the required number of grid points within inner part of the attached boundary layer in respect to Reynolds number [7]. This makes the classical LES approach improper for use in high Re number flows which appear in most of the industrial applications.

This fact stimulates attempts to apply and develop less expensive alternatives, such as LES modifications where the near wall regions are specially treated. One of the promising approaches for engineering applications is LES combined with near wall modeling (extensive review of such methods see in [7]). The approach has two important benefits: 1) it allows to use automatic mesh generation tools (to build meshes such as octree or mosaic); 2) due to near wall treatment
the meshes within the wall can be substantially coarser than in classical LES which results in significant decreasing of grid size.

The family of near wall models in LES is large and a number of simple and sophisticated methods exists [8–18], however they are typically calibrated and tested using canonical flows with attached boundary layer flows (flat plate, channel flow etc). In the current study we focus on a LES with classical wall modeling approach [8,11,19] here called as LES with Wall Function (WFLES). As shown in [20], when applied to problems with attached boundary layers good solution accuracy is obtained, however application to industrial type of the flows (with complex flow topology with physical effects such as flow separation and reattachment) is still rather questionable. Knowing the advantages of LES with near wall modeling listed above, defining its potential and limits in realistic flow configurations is important for engineering applications.

In this work we demonstrate capabilities of WFLES for turbulent flow around a simplified car body (known as Ahmed car) [21,22]. The configuration with 25° slant angle is considered, which provides relatively complex flow topology with boundary layer separation and reattachment above the slant. It is known that simulation the flow with LES or hybrid RANS-LES approaches requires substantially large computational resources. Therefore in literature simulations at lower Re number are considered (see e.g. [23, 24]), or computational results are sensitive to model, mesh or employed numerical method, see e.g. [25]. The goal of the current study is to justify the capabilities of WFLES to capture such a complex flow phenomena. The obtained results are compared with experiment data of [21] and [22].

2. Numerical setup
Simulation of flow around Ahmed body is carried out using incompressible branch of general purpose CFD solver ANSYS Fluent R21.1 [26] at \( Re = 2.8 \cdot 10^6 \) based on the body length, \( L_0 \). Ahmed car is positioned in a domain of size in stream-, span- and vertical- directions, respectively, equal to [-6,10], [-3,3] and [0,3.5] with the end of the body positioned at \( x = 0 \), see Figure (1). Symmetry BC is imposed on top, left and right boundaries of the domain; uniform inlet BC is imposed at inlet and pressure outlet BC with static (gauge) pressure is imposed at the outlet.

![Figure 1. Domain and body configuration in WFLES of Ahmed body car](image)

Thee meshes of octree type each of 7, 35 and 100 mln cells are used for mesh sensitivity study. Cubic cells within the attached boundary layer are used. Example of mesh resolution is shown on Figure (2) and the details on mesh resolution are listed in Table 1. WFLES is used with WALE turbulence model [5] with coefficient 0.5. The near wall treatment is based on classical wall function similar to [19]. The governing equations for the incompressible fluid are solved in the transient formulation by means of the finite volume method on collocated grids. Modification of the BCD scheme [27], namely the Weakly Bounded Central Difference
Figure 2. Mesh resolution over the back side of the body at middle XZ plane.

Table 1. Mesh specifications in WFLES of Ahmed body. The sizes of the mesh cells on top surface the body are given.

| Mesh   | $y^+_1 x=-0.5$ | $\delta/\Delta x=-0.5$ | Total size |
|--------|----------------|-------------------------|------------|
| Mesh-1 | 100            | 4                       | 7 mln      |
| Mesh-2 | 55             | 7                       | 35 mln     |
| Mesh-3 | 30             | 13                      | 100 mln    |

(WBCD), is employed. According to the author experience the scheme is robust and accurate for LES simulation of both wall bounded and free shear problems. The pressure gradient is discretised using a second order pressure scheme; gradient terms are approximated using the Least Square Cell Based (LSCB) scheme. SIMPLEC [28] 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 [29]. The time step $\Delta t$ is set such that the solution is time step independent. All simulations were performed in two stages. During a first transient period a statistically mature solution was obtained. After that averaging of the solution was started. Each of the periods were set to 5 flow passes over the body length.

3. Results
As seen on Figure (3) mesh refinement from Mesh-1 to Mesh-3 results in higher resolution of resolved turbulent structures, as expected. At the same time flow topology in the separation zone is independent on the grid used. This flow topology is in a good agreement with the experiments as shown at different cross sections on Figure (4).

Figure 3. Instant vorticity magnitude field on (from left to right) Mesh-1, Mesh-2 and Mesh-3
Figure 4. Flow comparison with experiment: a) streamlines over the slant in the middle XZ plane; cross flow distribution at downstream sections b) x=0.077 and c) x=0.190

Computed and experimental lengths of the recirculation zone as well as shape and position of the vortices are in a good agreement. Frequency of the vortex shedding behind the body is mesh independent as seen from the velocity spectra at a control point behind the body, see Figure (5).

Figure 5. Velocity spectra at a control point behind the body

Thus, the prediction of the main flow properties is sufficient for engendering applications even with the use of Mesh 1 of 7 mln of cells. Another situation is observed with more sensitive
flow details such as flow above the slant. As known from the experimental data, the flow in this region is characterized by a flow separation and reattachment. However, current numerical results in the region (see Figure (6)) depend on the mesh used and all the simulations are not in agreement with experiment. Namely almost no separation is predicted and, as a result, level of resolved kinetic energy is significantly underestimated.

![Figure 6](image)

**Figure 6.** Mean profiles of a) velocity and b) kinetic energy

As was mentioned in other WFLES studies, see e.g. [15], the reason of inaccurate flow field prediction with WFLES can be twofold: 1) lack of mesh resolution in the near wall areas with high gradients or 2) modeling errors of wall function approach employed. Note that Mesh 3 satisfies the minimum requirements for WFLES [20], namely it has cubic cells and the cell size is less than 10% of boundary layer thickness. However, near the slant beginning flow acceleration is observed which leads to boundary layer thinning. In this area even Mesh 3 becomes insufficient (see Figure (7)) and, as a result, velocity profile is predicted with significant error. So, for the prediction of the separation above the slant further mesh refinement is required. However, one may note that this would result in mesh size of hundreds of million cells, which are comparable with meshes for LES approach and are inappropriate for engineering applications.

![Figure 7](image)

**Figure 7.** Mean velocity profiles at $x= -0.262$, $x= -0.203$ and $x= -0.163$

4. Conclusions
WFLES simulations of the flow around Ahmed car were performed on three meshes with different resolution and size from 7 to 100 mln of cells. It is shown that the main flow structures are captures even on coarsest mesh considered. However more sensitive flow characteristics, such as separation and reattachment above the slant are not predicted even on the finest mesh of 100
mln of cells. It is shown that this mesh is insufficient for resolution of thin boundary layer before separation, so prediction of separation above the slant requires significant mesh refinement even in the frame of WFLES.

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).

References

[1] Spalart P R 1997 Comments on the feasibility of les for wings, and on a hybrid rans/les approach Proceedings of first AFSOR international conference on DNS/LES (Greyden Press)
[2] Strelets M 2001 Detached eddy simulation of massively separated flows 39th Aerospace sciences meeting and exhibit p 879
[3] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 International Journal of Heat and Fluid Flow 29 1638–1649
[4] Lilly D K 1992 Physics of Fluids A: Fluid Dynamics 4 633–635
[5] Nicoud F and Ducros F 1999 Flow, turbulence and Combustion 62 183–200
[6] Germano M, Piomelli U, Moin P and Cabot W H 1991 Physics of Fluids A: Fluid Dynamics 3 1760–1765
[7] Piomelli U and Balaras E 2002 Annual review of fluid mechanics 34 349–374
[8] Launder B and Spalding D 1974 The numerical computation of turbulent flows
[9] Deardorff J W et al. 1970 J. Fluid Mech 41 453–480
[10] Schumann U 1975 Journal of computational physics 18 376–404
[11] 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
[12] Hoffmann G and Benocci C 1995 Approximate wall boundary conditions for large eddy simulations Advances in turbulence V (Springer) pp 222–228
[13] Wang M 1999 Annual Research Briefs 355–364
[14] Cabot W and Moin P 2000 Flow, Turbulence and Combustion 63 269–291
[15] Wang M, Catalano P and Iaccarino G 2001 Annual Research Briefs 2001 45
[16] Jaegle F, Cabriot O, Mendez S and Poinsot T 2010 Flow, turbulence and combustion 85 245–272
[17] Bose S T and Moin P 2014 Physics of Fluids 26 015104
[18] Ambo K, Nagaoka H, Phillips D A, Ivey C, Brès G A and Bose S T 2020 Aerodynamic force prediction of the laminar to turbulent flow transition around the front bumper of the vehicle using dynamic-slip wall model les AIAA Scitech 2020 Forum p 0036
[19] Kader B 1981 International journal of heat and mass transfer 24 1541–1544
[20] Kolmogorov D, Menter F and Garbaruk A On mesh requirements for large eddy simulation with wall functions (submitted for publication)
[21] Ahmed S R, Ramm G and Faltin G 1984 SAE Transactions 473–503
[22] Lienhart H, Stoots C and Becker S 2002 Flow and turbulence structures in the wake of a simplified car model (ahmed model) New results in numerical and experimental fluid mechanics III (Springer) pp 323–330
[23] Krajnovic ’ S and Davidson L 2004 Large eddy simulation of the flow around an ahmed body Heat Transfer Summer Conference vol 46911 pp 653–662
[24] Krajnovic ’ S and Davidson L 2005
[25] Guilmineau E, Deng G, Leroyer A, Queutey P, Visonneau M and Wackers J 2018 Computers & Fluids 176 302–319
[26] Mathur S and Murthy J 1997 Numerical heat transfer 31 195–215
[27] Jasak H, Weller H and Gosman A 1999 International journal for numerical methods in fluids 31 431–449
[28] Van Doornmaal J and Raithby G 1984 Numerical heat transfer 7 147–163
[29] Rhie C and Chow W 1983 AIAA journal 21 1525–1532