

Validation of WMLES on a periodic channel flow featuring adverse/favorable pressure gradients

C. Carton de Wiart*, J. Larsson** and S. M. Murman*
Corresponding author: corentin.cartondewiart@nasa.gov

* NASA Ames Research Center, Moffett Field, CA, USA.

** University of Maryland, College Park, MD, USA.

Abstract: This study attempts at defining a new benchmark for the validation of wall models for wall-bounded turbulent flows in the presence of non-equilibrium effects. The case is designed to simulate non-equilibrium turbulent boundary layers while remaining accessible by most of the wall-modeling community and easy to setup. It is based on the turbulent channel flow, where the forcing in the momentum equation is modified to add non-equilibrium effects. To this end, a statistically converged solution using the equilibrium forcing is used as an initial solution and the simulation is restarted using different pressure gradients (adverse, transverse, favorable or a combination of these effects). The transient flow resulting from the sudden change of pressure gradient is then analyzed and space-averaged statistics are extracted. A set of very fine wall-resolved large-eddy simulations is first performed using five different pressure gradients: adverse, adverse/transverse, transverse, transverse/favorable and favorable pressure gradients. An equilibrium wall model is then applied on the case using a coarser grid. The results show that the equilibrium wall model is able to capture accurately the transient velocity profile but lead to some errors in the approximated friction magnitude and angle with the initial streamwise direction. Based on these preliminary results, the case seems to be a good candidate to provide a better understanding on the current wall modeling approaches and how we could improve them. Many parameters still need to be adjusted, such as the magnitude of the pressure gradient and the duration of the transient.

Keywords: Wall model, large-eddy simulation, non-equilibrium turbulent boundary layer, channel flow.

1 Introduction

Most industrial applications for Computational Fluid Dynamics (CFD) currently rely on Reynolds-Averaged Navier-Stokes (RANS) equations using flexible low-order methods. These approaches are accurate for high-Reynolds number flows with attached boundary layers, but fail to capture separated flows or transition from laminar to turbulent flow. As such, standard RANS methods are no longer sufficient to answer the design requirements of industry, in particular when off-design performance needs to be predicted. An alternative is to resolve the dynamics of the flow using unsteady techniques, such as Large Eddy Simulation (LES) or even Direct Numerical Simulation (DNS). In these “scale-resolving” simulations the full turbulent spectrum, or at least a significant portion, is resolved. Intuitively, with the continuous increase of computational power, one could say that scale-resolving simulations such as LES or DNS could now address this issue. Unfortunately, the cost of such simulations increases exponentially with the Reynolds number, making it impractical for industrial applications occurring at high Reynolds numbers. This is especially true for wall-bounded flows. As first pointed out by Chapman [1], and more recently by Choi and Moin [2], the number of grid points necessary to capture boundary layer turbulence using LES criterion scales with $Re^{13/7}$ ($Re^{37/14}$ for DNS). The increase of computational power and the recent advances in the industrialization of novel high-order methods allow scale-resolving simulations to be affordable for some moderate Reynolds number industrial

applications (turbomachinery, wind turbine, UAV, etc.). Nevertheless, most external aerodynamic targets are out of reach for the foreseeable future. For these high Reynolds number wall-bounded flows, modeling the near-wall region is necessary in order to make industrial computations practical.

As shown in the recent review of Larsson *et al.* [3], numerous wall models, mostly based on a wall-stress model, have been developed. In a wall-stress modeling approach, the inner part of the boundary layer is not computed but replaced by an equivalent momentum exchange with the wall through a boundary condition. The flow information away from the wall is used to evaluate the wall friction τ_w using an empirical wall model. The wall friction is then applied as a Neumann boundary condition where the velocity is free to fluctuate along the wall but not to penetrate it. The accuracy of the model can be adapted to the complexity of the case, as many different models can be found in the literature [4]. Analytic laws, based for instance on the simple logarithmic law, are easy to implement and give reasonable results on simple flows without non-equilibrium effects. It is unclear how these equilibrium models perform in the presence of strong pressure gradients. On the other hand, more expensive models, based on *Thin Boundary Layer (TBL)* approximation of the Navier-Stokes equations, are supposed to take into account those non-equilibrium effects to increase the accuracy of the wall model [5]. The effects of adverse pressure gradient and compressibility for example, can be taken into account in the TBL models and algebraic wall laws, leading to a growing universal validity with growing complexity of the model. More recently, a new type of wall-stress models, called integral wall models, have been studied in the literature. The integral wall model solves vertically-integrated momentum equations, capturing non-equilibrium effects to predict a more accurate wall shear-stress. Encouraging results using integral wall models have been obtained by Yang *et al.* [6] and Chung and Pullin [7] on non-equilibrium flows.

Although many different wall models have been developed and studied, it is still hard to discriminate the models using the current suite of benchmarks available in the literature. Most of these models perform well on equilibrium cases, typically on the turbulent channel flow, but the advantages of more complex wall models including non-equilibrium effects are unclear on more advanced cases [8]. Those more advanced cases are often too complex to find the source of errors or understand the impact of the wall model formulation, as shown in Carton de Wiart *et al.* [9]. Indeed, these cases often feature the combination of complex physics, such as transition from laminar to turbulent flow, favorable or adverse pressure gradient, separation and reattachment, *etc.* Most of these cases have been designed for the assessment of RANS approaches using experimental data as a reference, which also increases the level of uncertainty as it is difficult to reproduce exactly an experimental setup. From these results in the literature, it is hard to tell if the community should focus on the wall model formulation, the interaction of the wall model with the flow solver, the position of the wall model inputs or to the mesh requirements.

We believe a better-defined suite of benchmarks should be studied in order to answer these questions. To this end, it is convenient to define a list of benchmarks with growing complexity, starting from equilibrium flows and then adding progressively more complex physics, such as adverse/favorable/transverse pressure gradient, separation and reattachment, transition, complex physics, *etc.* For instance:

- Level-1: equilibrium flows, fully turbulent (e.g. channel flow, flat plate, etc.);
- Level-2: non-equilibrium turbulent boundary layer, no separation;
- Level-3: 2D-separation from fully developed, equilibrium turbulent boundary layer (e.g. curved backward facing step, boundary layer with blowing and suction on the top wall, etc.);
- Level-4: 2D-separation, preceded by an accelerated turbulent boundary layer (e.g. NASA 2D hump, 2D periodic hill, etc.);
- Level-5: full 3D flow, including separation and reattachment, transition, shocks, complex physics, etc.

Note that this is far from being an universal list of benchmarks. One could sort them in a completely different way and some of the levels could be split or merged in the future. Although arbitrary, defining these different levels helps to visualize the growing complexity of these benchmarks and therefore highlights the missing gap in the literature. Indeed, most of the time, studies jump from the well defined level-1 benchmark, featuring an equilibrium boundary layer, directly to a level-4 or 5 case, including complex non-equilibrium effects and flow separation, where it is hard to draw conclusion as all the non-equilibrium effects are combined.

In this study, a low level of complexity benchmark, a level-2 case, is defined by adding non-equilibrium effects to an equilibrium flow. The turbulence channel flow, as studied by Moser *et al.* [10], in which we modify the forcing term to generate transient non-equilibrium effects, is considered. Similar studies can be found in the literature. For instance, Seddighi *et al.* performed the DNS of a linearly accelerating channel flow [11] showing the effects of a favorable pressure gradients on the flow structure at the near-wall. On the same case, He *et al.* [12] studied the laminarization of flows at low Reynolds number. More recently, Giometto *et al.* [13] compared equilibrium and non-equilibrium wall models on a similar transient channel flow using a transverse pressure gradient. The case is appealing as favorable, adverse or transverse pressure gradient can be added one by one in order to study each non-equilibrium effect separately, or combined, and apply the wall models to observe the results. In this preliminary study, the pressure gradient is arbitrary set to obtain an unitary Clauser parameter. The duration of the transient is also set arbitrary to 20 convective times. These parameters of the simulation and their impact on the results will be analyzed in a near future.

All the simulations are performed using the compressible flow solver HybridX based on a sixth-order accurate finite difference method in space and a fourth-order accurate Runge-Kutta method in time. More details about the flow solver can found in the work of Larsson and Lele [14].

The first section presents the numerical setup and defines how the forcing that generates the non-equilibrium effects is applied. The second section discusses the flow physics resulting from the different non-equilibrium effects. Finally, the third section shows first results obtained on the case obtained using an equilibrium wall model.

2 Numerical setup

The case considered is the simulation of a periodic turbulent channel flow between two parallel walls, separated by a distance 2δ [10, 15]. The size of the domain is $4\pi\delta$ in the x-direction and $2\pi\delta$ in the z-direction. The flow is periodic in the wall-parallel directions. The flow conditions are chosen to match those of the reference DNS of Lee and Moser [15]. The friction Reynolds number is set as $Re_\tau = \rho u_\tau \delta / \mu = 1000$, with $u_\tau = \sqrt{\tau_w / \rho}$ being the friction velocity based on the wall shear stress τ_w . The friction Reynolds number is imposed through a constant forcing in the x-momentum equation using the pressure gradient $\frac{dp}{dx} = -\frac{\tau_w}{\delta}$. The bulk Reynolds number is defined as $Re_b = \rho u_b \delta / \mu$ and is a result of the simulation. According to the results obtained by Lee and Moser [15], the bulk Reynolds number should converge to $Re_b = 20000$. Using this value of Re_b , the Mach number is set to $M = u_b / c = 0.2$, with c the speed of sound, in order to approach an incompressible state.

Once the flow reaches a statistically steady equilibrium state, the simulation is restarted using a different pressure gradient vector in order to generate non-equilibrium effects. The value of the pressure gradient is set to obtain an unitary Clauser parameter

$$\beta = \frac{dp}{dn} \frac{\delta^*}{\tau_w} = 1, \quad (1)$$

using $\delta^* = 0.1152\delta$ the displacement thickness obtained by Lee and Moser [15] at $Re_\tau = 1000$. The initial forcing should be multiplied by 8.6790 to obtain a $\beta = 1$ and generate the transient non-equilibrium effects. As illustrated on Figure (1), five different non-equilibrium effects are studied: adverse pressure gradient, favorable pressure gradient, cross-flow pressure gradient (dp/dz) and two mixed adverse/favorable with transverse pressure gradients. The sudden change of pressure gradient magnitude (and direction) will accelerate or decelerate the flow until it reaches a new equilibrium state. The study will focus on the initial transient, in which the flow is in a non-equilibrium state. Instantaneous results are averaged in the spatial homogeneous directions and over the top and the bottom halves of the channel and are noted $\langle . \rangle$. Ensemble averaging using non-correlated initial flow solutions is not performed in this preliminary work but will be considered for future work.

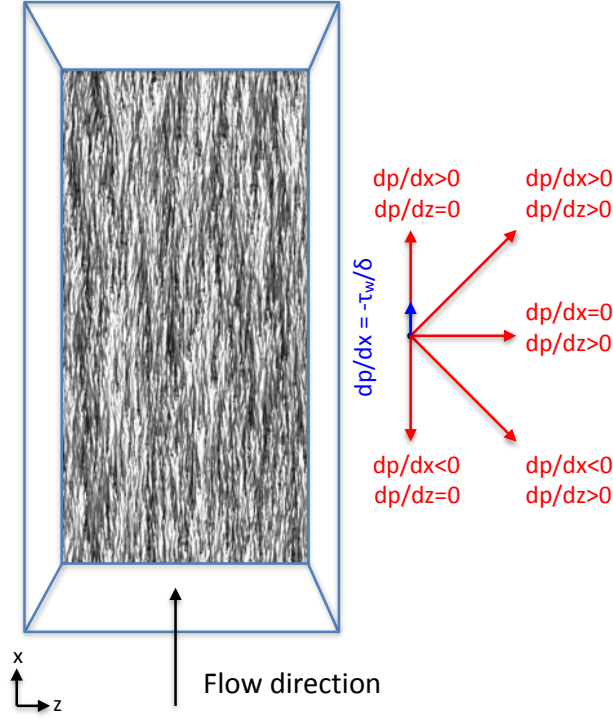


Figure 1: Illustration of the channel flow and the different non-equilibrium effects studied. A top view of the channel flow is presented together with the forcing vectors applied in this study. The blue vector represent the equilibrium forcing applied to generate the initial solution. The red vectors represent the different pressure gradients considered to generate the non-equilibrium effects.

3 Flow physics

A very fine Wall Resolved LES (WRLES) is performed on a $1536 \times 320 \times 768$ grid, resulting in a resolution of $8.2 \times 0.6 \times 8.2$ in wall units based on the initial wall friction. After obtaining a statistically steady equilibrium solution, five different simulations are performed, each case corresponding to a non-equilibrium effect. For all the non-equilibrium effects, an initial transient of $\Delta t_b = tu_b/\delta = 20$ is simulated, during which we collect spatially averaged statistics. Figure (2) shows an example of evolution for the velocity profile, here in the case with an adverse pressure gradient combined with a transverse component. It shows how the streamwise velocity decreases in time while the transverse velocity increases. At the final time, the velocity components have the same order of magnitude but have a different shape. Accelerating the boundary layer in the transverse direction does not compensate the decrease of velocity from the adverse pressure gradient, resulting in a velocity norm below the initial values. Similar analysis can be drawn from the other cases. For the validation of wall models, we are interested in how the velocity profile is close to the universal channel flow profile and how it deviates from the log-law. This can only be achieved by scaling the profiles using the instantaneous friction velocity u_τ . Figure (3) shows the evolution of the average velocity profile $\langle u \rangle^+ = \langle u \rangle / u_\tau$, where u_τ is in this case the instantaneous friction velocity magnitude and $\langle u \rangle$ the spatially-averaged norm of the velocity. Although the profile is close to the logarithmic law, some deviations can be observed. Accelerating the boundary layer seems to bring the velocity profile closer to the log-law at the center of the channel, removing the typical bump in the profile in the outer layer. Decelerating the boundary layer results in the opposite effect, the bump in the outer layer is increased and propagates towards the wall, creating a deviation from the log-law in the second part of the outer layer. A pure favorable pressure gradient results in a slightly higher position of the profile in the logarithmic region while it gives slightly lower values in the case of a pure transverse pressure gradient. These offsets in the logarithmic region cancel out when adding the two non-equilibrium effects together. The major deviations from the log law are observed when

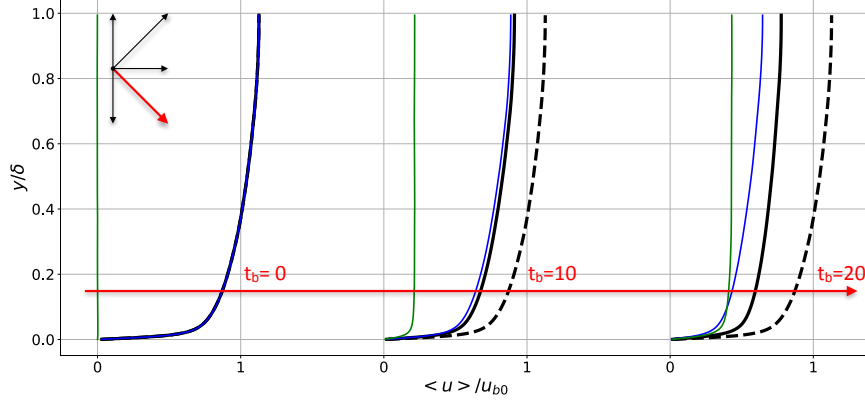


Figure 2: Case with transverse and adverse pressure gradients. Temporal evolution of the average velocity profile $\langle u \rangle / u_b$ at $t_b = 0, 10$ and 20 . Curves are shifted horizontally for clarity. Solid line: instantaneous velocity norm. Dashed lines: initial velocity norm. Blue line: streamwise velocity. Green line: spanwise velocity.

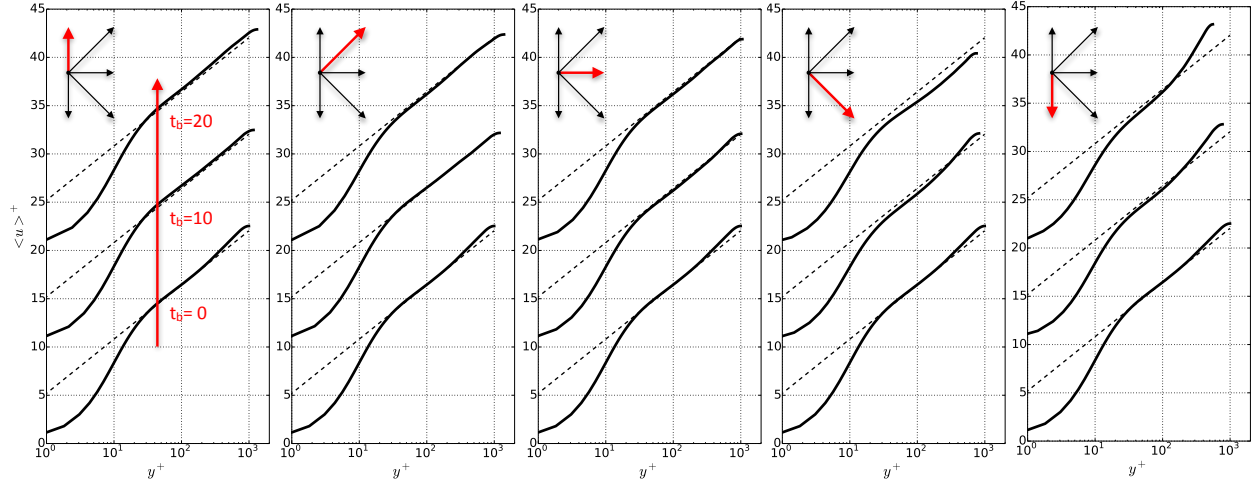


Figure 3: Temporal evolution of the average velocity profile $\langle u \rangle^+$ at $t_b = 0, 10$ and 20 . Curves are shifted vertically for clarity. From left to right: favorable, favorable+transverse, transverse, adverse+transverse and adverse pressure gradients. Solid line: WRLES. Dashed lines: logarithmic law.

an adverse pressure gradients is applied, especially when combined with a transverse component.

4 Results using an equilibrium wall model

The equilibrium wall model developed by Kawai and Larsson [16], based on simplified momentum and total energy equations, is then applied to the cases on a coarser grid containing $256 \times 96 \times 128$ degrees of freedom per equation. The input of the wall model is taken at a constant height $h = 0.1\delta$.

Figure (4) presents the temporal evolution of the velocity norm profile scaled with the instantaneous velocity friction extracted from the WRLES. For all the non-equilibrium effects, it can be seen that the velocity profiles match very well the WRLES results. Even for the pure adverse pressure gradient, which present the larger deviation from the equilibrium profile, the results obtained using the equilibrium wall model are very close to the WRLES.

Figure (5) shows the temporal evolution of the friction at the wall. Although the results are close to the reference WRLES, the equilibrium wall model does not consistently capture the temporal evolution of the

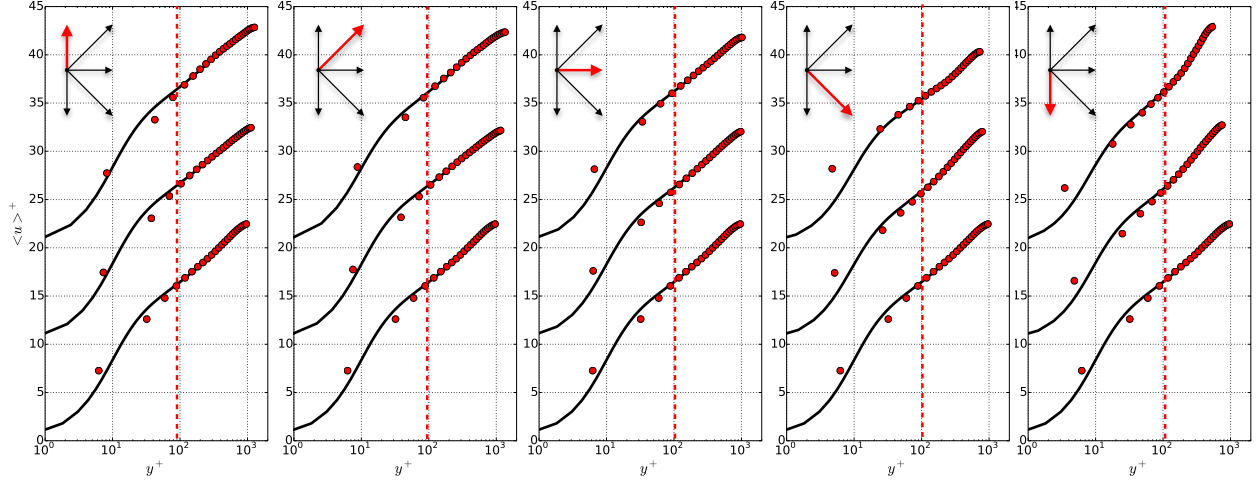


Figure 4: Temporal evolution of the average velocity profile $\langle u \rangle^+$ at $t_b = 0, 10$ and 20 . Curves are shifted vertically for clarity. From left to right: favorable, favorable+transverse, transverse, adverse+transverse and adverse pressure gradients. Black solid line: very fine LES. Red circles: equilibrium WMLES. Vertical dashed lines indicate the position of the wall model inputs.

wall friction. As shown on Figure (6), this is due to the fact that the equilibrium wall model always assumes

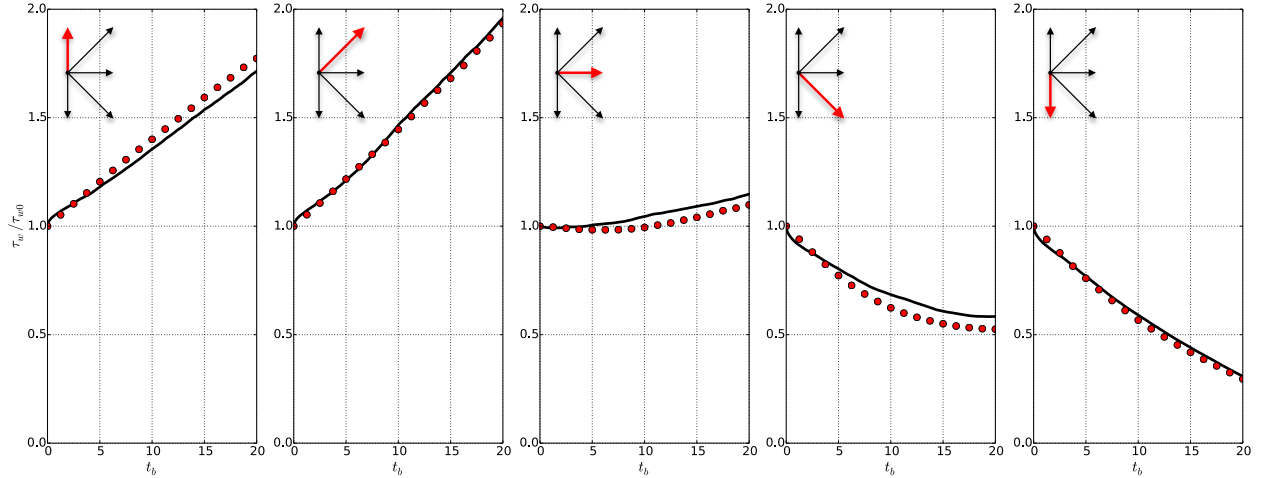


Figure 5: Temporal evolution of the wall friction τ_w . From left to right: favorable, favorable+transverse, transverse, adverse+transverse and adverse pressure gradients. Black solid line: WRLES. Red circles: equilibrium WMLES.

that the velocity profile lies on the log-law. As the non-equilibrium effects deviates the velocity profile from the log-law, it generates an error in the wall friction computed from the equilibrium wall model. Even on the cases where the error on the friction is significant (10% on the favorable/transverse pressure gradient), the velocity profile is still well captured, as shown in Figure (4). This indicates that the error on the friction does not have the time to be propagated back to the turbulence, and does not impact significantly the velocity profile. Running the case longer in time should give us more clues on how a mismatch in the friction can affect the turbulence in the long term.

Finally, the three-dimensional effects of adding a transverse component to the two-dimensional system is studied. As shown on Figure (7), a transverse pressure gradient will twist the boundary layer profile, resulting in an averaged velocity vector that is not aligned with the friction anymore. This breaks one of the

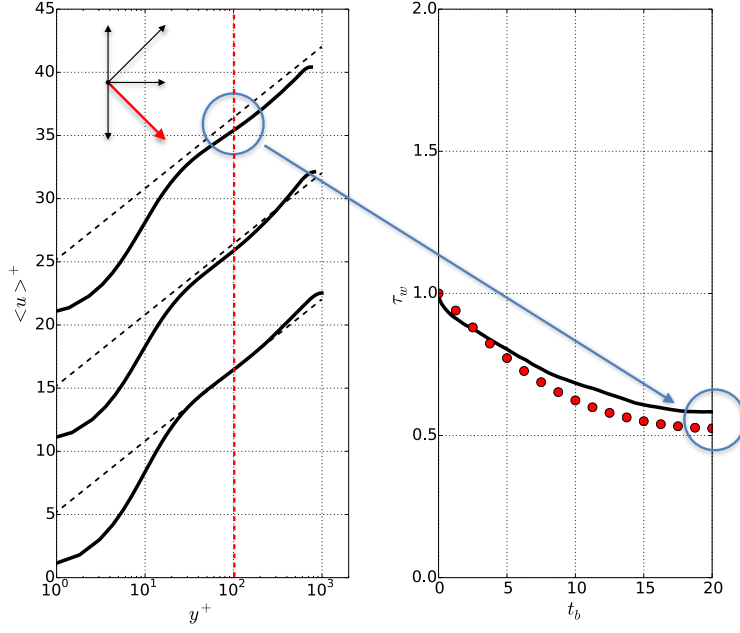


Figure 6: Adverse+transverse pressure gradient case. Illustration on how a deviation from the log-law in the velocity profile gives an error in the friction computed from the equilibrium wall model.

main hypothesis of most wall-stress models, which assume the friction is aligned with the velocity.

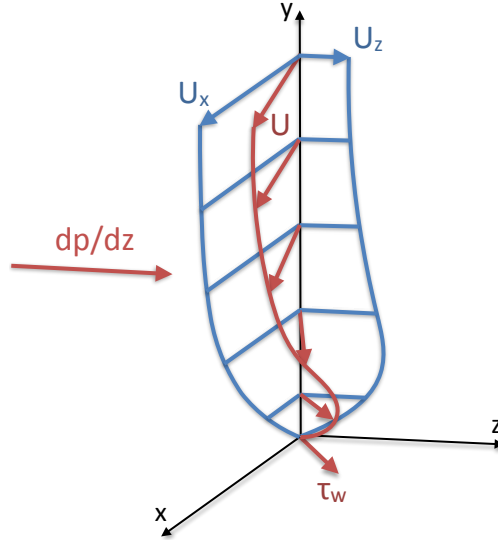


Figure 7: Illustration of the twisting of the velocity profile in a boundary layer with transverse pressure gradient.

Figure (8) shows the temporal evolution of the angle between the wall friction vector and the x -axis (aligned with the initial streamwise direction) for the three cases with transverse pressure gradients. Both WRLES and WMLES results are presented, together with the deviation of the velocity vector extracted at $y/\delta = 0.1$. As mentioned before, the equilibrium wall model assumes that the friction is aligned with the velocity. Therefore, the velocity and the wall friction deviations are exactly the same in the WMLES case. Due to this restriction of the WMLES approach, it fails to capture the correct deviation of the wall friction vector. The friction stays aligned with the velocity, giving a maximum error of 10° in the case with adverse

and transverse pressure gradient.

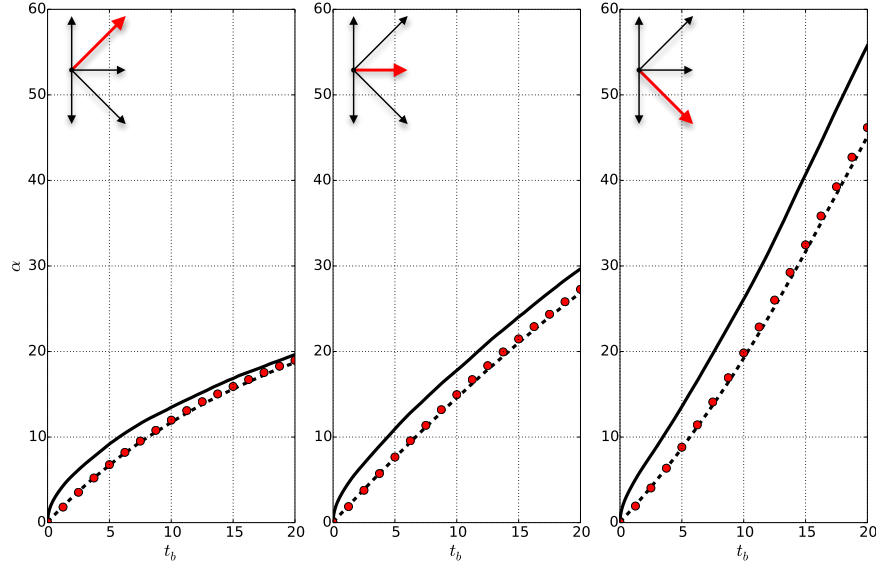


Figure 8: Temporal evolution of the deviation of the wall friction vector from the x-axis. From left to right: favorable+transverse, transverse and adverse+transverse pressure gradients. Black solid line: WRLES. Red circles: equilibrium WMLES. Velocity deviation from the WRLES extracted at $y/\delta = 0.1$ is also shown using black dashed line.

5 Conclusion and future work

We started to define a more rigorous validation test suite of WMLES based on numerical experiments. The ultimate goal of the work is to generate a database that could be used by the WMLES community to validate their wall model approaches. A first benchmark for the validation of wall models for flows featuring adverse, favorable or transverse pressure gradient has been defined. A very fine WRLES has been run on the case as well as an equilibrium wall model on a coarser mesh. The results show that, even if the wall model does not predict the correct temporal evolution of the wall friction, we can still capture the non-equilibrium effects in the velocity profile. This means that, in this particular case, the errors in the friction are not directly propagated back into the velocity profile. The time scale of the forcing is probably too small regarding that of the turbulent response. The case also shows that a transverse pressure gradient breaks the hypothesis of the equilibrium wall model in which the wall friction is supposed to be aligned with the velocity at the input location. Finally, the case can be used for *a priori* and *a posteriori* validation of wall model approaches.

These results are preliminary and many improvements can be done on the case. Firstly, ensemble averaging should be performed in order to be able to extract higher order statistics, which need more sampling in order to be statistically converged. The impact of the magnitude of the forcing on the results should be analyzed and the value of the Clauser parameter should be linked to more realistic benchmarks in the literature (NASA 2D hump, 2D periodic hill, *etc.*). A longer transient should be considered in order to see if an error on the wall friction can ultimately have a significant impact on the results or if capturing the non-equilibrium effects in the outer layer is sufficient to capture the velocity profile. More advanced wall models including non-equilibrium effects will also be applied on the case. As a first step, the WRLES results will be used to perform an *a priori* test of different wall models, including non-equilibrium effects.

The next step of the validation would be to define a case with the next level of complexity, featuring a 2D-separation from a fully developed turbulent boundary layer profile, *i.e.* without accelerating the boundary layer. Several interesting cases have been identified, including that in the recent study of Coleman *et al.* [17], where a separation bubble is generated through a suction and blowing boundary condition.

References

- [1] D.R. Chapman. Computational aerodynamics development and outlook. *AIAA Journal*, 17(12):1293–1313, 1979.
- [2] H. Choi and P. Moin. Grid-point requirements for large eddy simulation: Chapman’s estimates revisited. Annual research briefs, Center for Turbulence Research, 2011.
- [3] J. Larsson, S. Kawai, J. Bodart, and I. Bermejo-Moreno. Large eddy simulation with wall-modeled wall-stress: recent progress and future directions. *Mech. Eng. Reviews*, 3(15-00418), 2016.
- [4] S. Bocquet, P. Sagaut, and J. Jouhaud. A compressible wall model for large-eddy simulation with application to prediction of aerothermal quantities. *Physics of fluids*, (065103), 2012.
- [5] G. I. Park and P. Moin. Numerical aspects and implementation of a two-layer zonal wall model for les of compressible turbulent flows on unstructured meshes. *J. Comput. Phys.*, 305:589–603, 2016.
- [6] X. I. A. Yang, J. Sadique, R. Mittal, and C. Meneveau. Integral wall model for large eddy simulations of wall-bounded turbulent flows. *Physics of Fluids*, 27(2), 2015.
- [7] D. Chung and D. Pullin. Large-eddy simulation and wall modelling of turbulent channel. *J. Fluid Mech*, 631(281), 2009.
- [8] G. I. Park. Wall-modeled large-eddy simulation of a high Reynolds number separating and reattaching flow. *AIAA Journal*, 55(11), 2017.
- [9] C. Carton de Wiart and S. Murman. Assessment of wall-modeled LES strategies within a discontinuous-Galerkin spectral-element framework. In *55th AIAA Aerospace Sciences Meeting*, number 2017-1223, 2017.
- [10] R. D. Moser, J. Kim, and N. N. Mansour. Direct numerical simulation of turbulent channel flow up to $Re_\tau = 590$. *Physics of fluids*, 11(943), 1999.
- [11] M. Seddighi, S. He, A. E. Vardy, and P. Orlandi. Direct numerical simulation of an accelerating channel flow. *Flow Turbulence Combust*, 92(473), 2014.
- [12] S. He, K. He, and M. Seddighi. Laminarisation of flow at low Reynolds number due to streamwise body force. *J. Fluid Mech.*, 809:31–71, 2016.
- [13] M. G. Giometto, A. Lozano-Duran, G. I. Park, and P. Moin. Three-dimensional transient channel flow at moderate Reynolds numbers: Analysis and wall modeling. *Center for Turbulence Research Annual Research Briefs*, pages 193–205, 2017.
- [14] J. Larsson and S. K. Lele. Direct numerical simulation of canonical shock/turbulence interaction. *Physics of fluids*, 21(126101), 2009.
- [15] M. Lee and R. D. Moser. Direct numerical simulation of turbulent channel flow up to $Re_\tau = 5200$. *J. Fluid Mech*, 774:395–415, 2015.
- [16] S. Kawai and J. Larsson. Wall-modeling in large eddy simulation: Length scales, grid resolution, and accuracy. *Physics of fluids*, 24(015105), 2012.
- [17] G. N. Coleman, C. L. Rumsez, and P. R. Spalart. Numerical study of turbulent separation bubbles with varying pressure gradient and Reynolds number. *J. Fluid. Mech.*, 847:28–70, 2018.