WORKSHOP
Direct and Large-Eddy Simulation 14
April 10-12 2024, Erlangen, Germany

HIGH-FIDELITY SIMULATIONS TO ACCESS THE AERODYNAMIC PERFORMANCE OF REAL ROAD VEHICLES

S. Sebben[1]
[1]Department of Mechanical and Maritime Sciences
Chalmers University of Technology, Sweden
simone.sebben@chalmers.se

INTRODUCTION

Governmental legislations and customers awareness of the negative effects of transportation on the global climate are driving the development of more energy efficient vehicles. One of the key factors influencing the energy efficiency is the aerodynamic drag. The importance of minimizing drag has increased with the event of electrification as the driving range is a limiting factor in most BEVs (Battery Electric Vehicles). However, low drag streamlined cars might impair instabilities which can be perceived by the customer as a vehicle nervous to drive. It is, therefore, important to both minimize the drag and the lift of the vehicle.

A significant body of work is available in the literature related to reducing vehicle drag by improving the rear wake using shape optimization or some form of active flow control. Many such investigations have been performed on simplified models without considering yaw conditions or driving stability issues. Few examples are [3, 4, 5]. This is because numerical simulations of more realistic situations are computational expensive if high-fidelity models are to be used. In this paper, numerical simulations using a hybrid turbulence model, the stress-blended eddy simulation (SBES), are conducted on a full-scale vehicle geometry at relevant Reynolds number and realistic driving conditions. The dynamics of the wake and the temporal fluctuations of the aerodynamic lift are analysed and coupled to the driver's perception of control and stability.

CFD METHODOLOGY

Numerical simulations with the commercial code Ansys Fluent are performed using a hybrid RANS-LES approach, often with the $\kappa - \omega$ SST model in the RANS region and the Smagorinsky model in the LES region. In most cases, the meshes are hexahedral dominated and unstructured with 10 to 15 prism layers on the vehicle surfaces and ground. Second order temporal (implicit) and spatial discretization schemes (hybrid bounded central difference /second order upwind) are used for all dependent variables.

The mesh and time dependence studies consider a compromise between lead time/CPU availability and accuracy. Regarding mesh accuracy, the following is evaluated: variation of global variables (drag and lift coefficients), amount of resolved kinetic energy, turbulent viscosity ration, blending factor, and two-point correlation in the wake, as suggested by Davidson [1]. The size of the time-step is checked by ensuring that the Courant number is one or less in most of the domain, although this is not achievable in areas with strong acceleration. So, to confirm its validity, a normalized value defined by Ekman et al. [2] as the number of time steps needed for the free stream to travel the length of the vehicle is considered. According to [2], a value above 400 is sufficient. This is always attained in all computations. Examples of how some of these properties are evaluated can be seen in Figure 1.

With the aforementioned considerations, typical mesh sizes range from 200 to 300 million cells and take between 30-100 thousand CPU hours.

RESULTS

In this work, numerical simulations of two variants of a rear roof spoiler on a real sports utility vehicle (SUV) are presented. Both spoilers had time-averaged lift coefficients within the set requirements, but the baseline spoiler showed lower straight-line stability performance at high speeds during previous on-road tests. A CFD analysis revealed the importance of considering the unsteady rear lift fluctuations, C'_{lr}, as these could be associated with frequencies prone to cause the vehicle instabilities observed on the road. Figure 2 shows that the baseline spoiler experiences higher amplitude of fluctuations in the sensitive frequency range for vehicle dynamics (0.5-2 Hz), particularly with yaw (only 5 deg shown here).

The unsteady vertical base pressure gradient was monitored on several positions at the rear and is plotted as probability density function (PDF) in Figure 3. It shows that the wake of the baseline spoiler (Figure 3a) has two states, here defined as H and L, representing the wake dynamics that generate high and low rear lift forces, respectively. The improved spoiler, Figure 3b, has only one state with a higher vertical base pressure which indicates more upwash, thus meaning lower rear lift. The lateral base pressure gradient was also calculated and showed one state for both spoilers.

Finally, the improved spoiler increased drag compared to the baseline for all flow angles. However, its sensitive to yawed flow was less, resulting in a flatter curve.

CONCLUSIONS

An analysis of the unsteady lift forces showed that low-frequency fluctuations, close to the first natural frequency of the rear suspension, were present for the baseline spoiler at yaw angles. The vertical base pressure gradient indicated the existence of a bi-stable wake which was connected to the driver's perception of instability during on-road tests. An improved spoiler was suggested and successfully solve this problem, however, to the cost of slightly higher drag.

REFERENCES

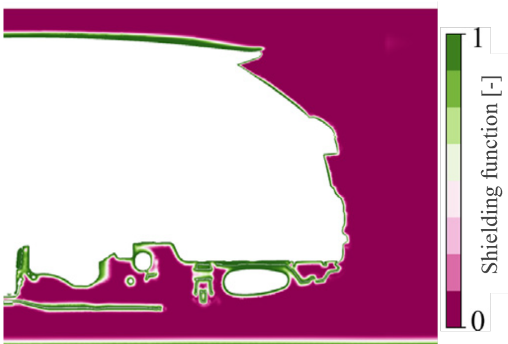
[1] Davidson, L. : Large eddy simulations: How to evaluate resolution, *Int. J. Heat Fluid Flow*, **30 (5)**, 1016–1025 (2009).

[2] Ekman, P., Larsson, T., Virdung, T. and Karlsson, M. : Accuracy and speed for scale-resolving simulations of the DrivAer reference model, *SAE Technical Paper 2019-01-0639, Detroit, USA*, (2019).

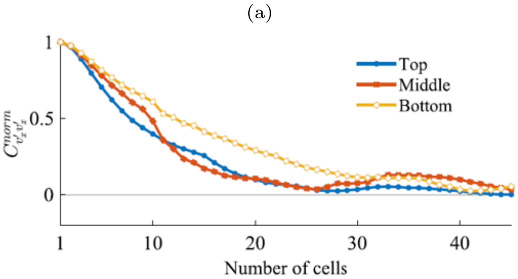
[3] Haffner, Y., Li, R., Meldi, M. and Borée, J. : Drag reduction of a square-back bluff body under constant cross-wind conditions using asymmetric shear layer forcing, *Int. J. Heat Fluid Flow*, **96**, 1016–1025 (2022).

[4] Podvin, B., Pellerin, S., Fraigneau, Y., Bonnavion, G. and Cadot, O. : Low-order modelling of the wake dynamics of an Ahmed body, *J. Fluid Mechanics*, **927** (2021).

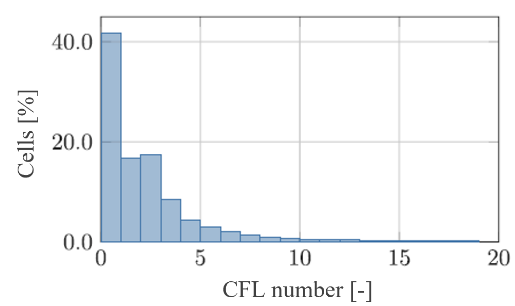
[5] Siniša, K. and Fernandes, J. : Numerical simulation of the flow around a simplified vehicle model with active flow control, *Int. J. Heat Fluid Flow*, **32 (1)**, 192–200 (2011).



(a)



(b)



(c)

Figure 1: (a) Blending function at y=0 indicating regions of LES (purple) and URANS (green), (b) Two-point correlation on three horizontal lines in the wake, and (c) Typical CFL distribution for all cells in the domain.

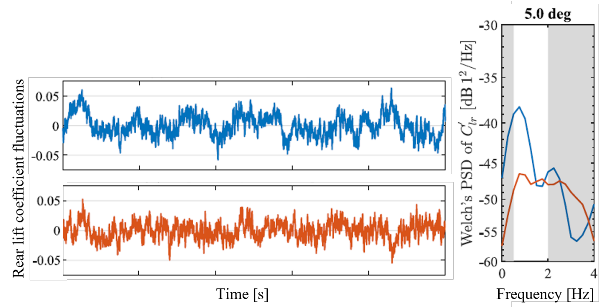
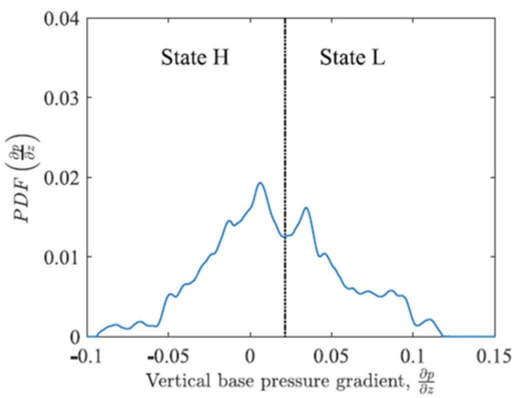
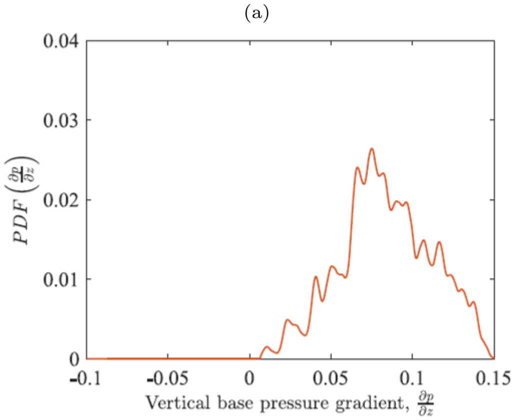


Figure 2: (Rear lift fluctuations for the baseline spoiler (blue) and the improved spoiler (red) for 5 deg flow angle, and Welch's PSD frequency response.



(a)



(b)

Figure 3: Probability density function of the vertical base pressure gradient at 5 deg: (a) Baseline spoiler and (b) Improved spoiler.