UPCommons

Portal del coneixement obert de la UPC

<http://upcommons.upc.edu/e-prints>

© 2018. Aquesta versió està disponible sota la llicència CC-BY-NC-ND 4.0 <http://creativecommons.org/licenses/by-nc-nd/4.0/>

© 2018. This version is made available under the CC-BY-NC-ND 4.0 license <http://creativecommons.org/licenses/by-nc-nd/4.0/>

Flow over a realistic car model: Wall modeled large eddy simulations assessment and unsteady effects

D.E. Aljure^a, J. Calafell^a, A. Baez^a, A. Oliva^{a,*}

^a*Center Tecnològic de Transferència de Calor (CTTC) , Universitat Politècnica de Catalunya (UPC). ETSEIAT, Colom 11, 08222, Terrassa, Barcelona, Spain. Fax: +34 93 739 89 20. cttc@cttc.upc.edu*

Abstract

Numerical simulations are carried out on the flow over a realistic generic car geometry, the DrivAer-fastback car model. Pure large eddy simulations (LES) and wall-modeled large eddy simulations (WMLES) are used and compared to numerical and experimental results to assess the validity of these approaches when solving the flow field around complex automotive geometries. Results show a 70% CPU time reduction when using the wall model. Drag coefficient results show the influence of the wall model on coarser meshes is positive, reducing the difference on those obtained using finer meshes. Pressure profiles exhibit mixed results. The wall model used works well in adverse pressure gradients and smooth geometry changes. Results worsen in sections where the flow detaches and experiences large pressure drops. Flow structures and unsteady effects around the car are also analyzed, obtaining several characteristic frequencies for the different flow structures encountered. It should be noted that the present investigation shows how WMLES helps to reduce

*Corresponding author

Email addresses: davidal@cttc.upc.edu (D.E. Aljure), joancs@cttc.upc.edu (J. Calafell), aleix@cttc.upc.edu (A. Baez), oliva@cttc.upc.edu (A. Oliva)

the computing cost and response vs pure LES, while providing high-quality unsteady data, although computational cost remains high. Results show potential in the introduction of this tool as a competitive simulation strategy for complex geometries.

Keywords: Automotive aerodynamics, CFD, wall model LES, Turbulence models

1. INTRODUCTION

Automotive aerodynamics have a direct and strong impact on fuel consumption, driving stability, and ride noise both inside and outside the cabin. The effect of fuel consumption due to automotive aerodynamics can be analyzed from the averaged drag forces extracted from time integrated pressure and velocities fields. On the other hand, unsteady data is required to analyze driving stability and aerodynamic noise. This imposes challenges when studying these phenomena with Computational Fluid Dynamics (CFD).

The use of CFD makes possible the study of a wide variety of turbulent flow cases, including geometries that present massive flow separations such as those encountered in a variety of industrial related applications. The automotive industry makes great advances every year; engine performance has increased greatly in the past decade, along with weight reduction and safety measures. These advances make aerodynamics ever more important, and as such, investigation on this subject has greatly increased. In this sense, numerous experimental and numerical research works have been carried out in the subject (Khalighi et al., 2012; Bello-Millán et al., 2016; Bonnavion et al., 2017; Lichtneger and Ruck, 2017).

The main restriction in the application of CFD in automotive aerodynamics lies in the computing cost and turn around times required to simulate steady and unsteady flows. The simulation of the flow around cars, aiming to predict lift and drag forces, has been conventionally carried out with the Reynolds Averaged Navier Stokes (RANS) approach. These simulations allow car makers to obtain reference values at industrially feasible computing costs and response times. However, this technique does not allow the study of unsteady phenomena, as well as failing to correctly predict the flow around bluff and blunt ended bodies such as hatchback, estate and SUV type cars.

Recent computational advances have made possible the study of very complex cases by using high performance computing (HPC) and advanced numerical algorithms, such as large eddy simulations (LES), detached eddy simulations (DES)(Spalart, 2009) or Lattice Boltzman based methods (Chen et al., 2003). These techniques are quickly entering into industrial research due to the necessity of correctly predicting unsteady flow phenomena. In particular, LES techniques are being the focus of many researchers in order to allow its introduction into industrial research, however, they still require large amounts of computational resources. In this simulations only the large scales of motion are explicitly solved, capturing most of the relevant unsteady phenomena, whereas diffusive scales are modeled. Recent advances divide the LES modeling strategy into two main branches. In classical LES a suitable sub-grid model has to be implemented don the solver in conjunction with preserving schemes. In implicit LES (Adams and Hickel, 2009), subgrid diffusive scales are modeled by means of numerical dissipation introduced by the discretization schemes.

One of the main limitations when using LES is the high computational cost to properly resolve the wall regions, from both a spatial and temporal viewpoint, since these areas enclose a wide range of different scales of motion. It is precisely this restriction what has been delaying the applicability of LES, and CFD in general, to automotive aerodynamics. The near wall region, i.e. boundary layer, is of fundamental importance in the overall flow field for bluff body aerodynamics. The boundary layer development influences several flow aspects, such as the separation points and shear layer formation, among others. These aspects change the overall wake topology and obtained force coefficient. Hence, a proper mesh resolution in this area is of vital importance to achieve accurate results. Since length scales decrease dramatically as the flow approaches the wall, especially at high Reynolds, very small control volumes are required in these regions, increasing the cell count and restricting the time step making LES of wall bounded flows at high Reynolds almost un-affordable. The wall modeled large-eddy simulation (WMLES) aim is to provide a suitable and cost-effective solution for wall bounded flows, resolving the larger eddies of the flow away from the wall and modeling the complex phenomena in the near-wall region. In WMLES, the grid resolution in the wall area is not enough to capture all the flow-related phenomena. Nonetheless, if the grid is large enough, the flow behavior encountered there can be treated as an ensemble average and statistical tools used (Piomelli and Balaras, 2002).

Different strategies have been developed in order to model the boundary layer, being the hybrid RANS-LES and wall shear stress models (WSSM) the most relevant. In the first case, the boundary layer region is modeled by

means of a RANS model, while the far field is solved with a LES approach. In the wall shear stress strategy, the whole domain is resolved with a LES model while an accurate wall shear stress, obtained with an external model, is supplied to the LES domain through the solid boundary. A wide variety of WSSM formulations can be found in the literature with different degrees of accuracy and ranges of applicability, namely wall functions (WF), integral boundary layer approaches and zonal or two layer models (TLM). In the simplest wall models (i.e. the wall functions), the wall shear stress is a function of the LES velocity field and it is evaluated through an algebraic expression derived from an a-priori known velocity field, such as in the method exposed by Werner and Wengle (1991). This model calculates the wall shear stress using a model based on the law of the wall. The integral boundary layer models could be considered a more accurate variant of the wall functions in which the governing equations are integrated in the wall normal direction considering non-equilibrium terms such as the advective and the pressure gradient (Yang et al., 2015). In the two layer models approach, governing equations are fully or numerically resolved in a fine embedded mesh which extends between the solid wall and a given height lower than the boundary layer thickness. The degree of accuracy is variable depending on the equations solved in the model which may range from a simple diffusive term (i.e. equilibrium model) to the full RANS equations (Kawai and Larsson, 2013; Park and Moin, 2014). An extensive review on hybrid RANS-LES approach was published by Spalart (2009), while for wall modeling in general and two layer models in particular, detailed information can be found in the reviews of Piomelli (2008) and Larsson et al. (2016), respectively.

The present work aims to the capabilities of WMLES in automotive aerodynamics, for which reference experimental data is required for the sake of comparison. Most CFD research on automotive external aerodynamics has been carried out using either, simplified models such as the Ahmed, Asmo or Windsor car models (Ahmed et al., 1984; Aljure et al., 2014b; Islam et al., 2017), or basic car shapes such as the MIRA reference car and its variants (Le Good and Garry, 2004). The simplified bodies allow researchers to study fundamental aerodynamic processes and focus on specific features of automotive aerodynamics. Basic car shapes have been mostly used for wind tunnel calibration and validation of CFD methodologies. As these geometries vastly differ from actual car geometries, the results exhibit a high degree of abstraction and do not fully reproduce the real aerodynamic behavior present in automotive geometries. To overcome this deficiency, the DrivAer car model was introduced in 2012 (Heft et al., 2012) based on the geometries of the Audi A4 and the BMW 3 series. Although it is a basic car shape, this realistic generic car body was conceived to investigate the specific flow structures present in current production cars.

Heft et al. (2012) performed wind tunnel experiments on different configurations of the DrivAer model. These configurations are divided by several features: type of back (fastback, estate back and notch back), underbody (detailed and simplified), mirrors (with and without), wheels (with and without) and ground simulation (with and without). The obtained results showed that the influence of the Reynolds number on the drag coefficient lessens with augmenting Reynolds numbers, being almost independent for $Re \geq 4.87 \times 10^6$. Furthermore, pressure profiles over the top, bottom and side of the car were

reported in this initial investigation. The comparison between the profiles obtained along the top of the vehicle with and without ground simulation showed nearly no difference, whereas a higher influence was observed on the bottom. Finally, they state the large influence the mirrors exert on the flow over the windshield and side windows, affecting the A-pillar vortex formation. Strangfeld et al. (2013) and Wieser et al. (2013) performed further experimental observations on the DrivAer geometry, including side wind conditions. Their results showed that, for the fastback configurations, independence of the drag coefficient with regards to Reynolds number was achieved at around $Re = 2 \times 10^6$, less than half the value reported by Heft et al. (2012). These experiments were carried out without a moving ground. Moreover, the force measurements were in very good agreement with those performed at TU-Munich by Heft et al. (2012). A decomposed pressure distribution of the drag and lift coefficients helped identify the dominant regions most affecting the forces around the car. For the no side-wind case, the areas identified were the upper region of the rear window and the c-pillars. Additionally, from their results a separation bubble was identified in the models rear (at the start of the rear window), showing a symmetric pattern for the Fastback configuration. They also observed considerable fluctuations within the car's wake, which could affect the force measurements. A spectral analysis performed showed several dominant frequencies in the flow, confined in the rear sections of the car and dependent on the measurement location.

Several numerical works have also been carried out with the DrivAer model, particularly RANS simulations (Shinde et al., 2013; Guilmineau, 2014a,b; Peters et al., 2015; Ashton et al., 2016; Jakirlic et al., 2014) and

hybrid approaches such as detached eddy simulations (DES) (Guilmineau, 2014a; Ashton et al., 2016) or partial averaged Navier-Stokes (PANS) (Jakirlic et al., 2014). Other numerical approaches have also been used, such as the lattice Boltzmann method (LBM) (Pasquali et al., 2015). These authors have used meshes ranging from 19×10^6 to 125×10^6 control volumes for Reynolds number $Re = 4.87 \times 10^6$. These authors point out the improvement of results obtained when using unsteady simulations, showing that these algorithms are more suitable to correctly predict forces and correctly capture flow phenomena in automotive external aerodynamics. The flow structure described by the different authors show a very complex wake which exhibits large recirculation regions behind the car (Ashton et al., 2016), counter rotating vortices stemming from the under body (Jakirlic et al., 2014) and vortices detaching from the A-pillars and the mirrors (Guilmineau, 2014b). It should be noticed that several investigations show that the flow remains attached above the rear window (Guilmineau, 2014b; Jakirlic et al., 2014), while the observations of Strangfeld et al. (2013) and Wieser et al. (2013) showed the opposite. In most works a general good agreement of the numerical pressure profiles with those obtained experimentally was reported, however some deviations were found, specially in the vehicle's top. Peters et al. (2015) pointed to the presence of the roof stinger in experiments and the differences between the CAD and real geometries as the source for these discrepancies. In addition, force coefficients show some scattering, specially in the lift coefficient. Ashton et al. (2016) identified the major source of error at the car's rear section, where separation regions are found and the flow structure is more complex.

Fewer papers have focused on the impact of the rotating wheels and

ground simulations on the overall flow configuration over the DrivAer car geometry. Guilmineau (2014b) identified the interactions present between the rotating wheels and the underbody of the car, leading to a reduction in both the drag and lift coefficient (with respect to the same geometry with stationary wheels). This change in force coefficients was also observed experimentally by Heft et al. (2012) and numerically by Shinde et al. (2013) among others. Other observed effects include a change in the vortices stemming from the wheels, particularly those forming around the front wheels. The pressure distribution along the top of the DrivAer fastback car does not seem to be very affected by the ground simulation, however, significant changes were observed in the pressure profiles around its side and bottom.

One of the main characteristics of external automotive aerodynamics is the wide variety of unsteady effects present in its flow configuration. In this sense, several authors have pointed out the existence of time-periodic phenomena in car-like geometries. Duell and George (1999), studying a square-back simple geometry, and Gilhome (2002) in a notch-back type geometry; both observed characteristic frequencies within the flow around the models and attributed them to large-scale and shear layer vortex shedding. This phenomena has also been observed by Wieser et al. (2013) in his experiments around the DrivAer fastback model. In fact, Wieser et al. (2013) characterized several frequencies around the DrivAer model. Near the model's base region (rear bumper area) a Strouhal number ($St = fH_{ref}/u_{ref}$) of $St \approx 0.07$ was detected (St number normalized with base height). This frequency coincides with the pumping effect described by Duell and George (1999). A second frequency component of $St \approx 1.27$ detected near the base edges is

argued to be a shedding frequency. Frequencies of $St = 0.21$ and $St = 4.63$ were also observed around the car's back. Wieser et al. (2013) related the former to the characteristic frequencies observed in generic bluff bodies, whereas the higher frequency was associated to the flow detaching from the top of the rear window. A frequency around $St \approx 0.28$ observed behind the vehicle was related to the rear wheels rotation.

Most of the numerical research work performed so far on the DrivAer fastback model has been carried out using steady state simulations while unsteady simulations have been limited to DES and PANS modeling approximations. In this sense, the present work aims to assess the WMLES approach for solving automotive aerodynamic type flows. Furthermore, we aim to advance in the understanding of the physics governing the flow over the DrivAer geometry, in particular the unsteady effects inherent in automotive aerodynamics. To do so, LES cases are carried out to assess mesh resolution and temporal convergence and integration. Afterwards, the WMLES strategy for automotive aerodynamic type flows will be assessed and evaluated by comparing the results obtained from the WMLES with a pure LES solution. Finally, an analysis into the unsteady effects and flow structures found around the car will take place.

2. Mathematical and Numerical Model

2.1. Governing equations

In order to study the flow, the filtered incompressible Navier-Stokes equations are solved numerically:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial(\overline{u}_i \overline{u}_j)}{\partial x_j} - \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} + \rho^{-1} \frac{\partial \overline{p}}{\partial x_i} = - \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

where $\overline{u}_i$ stands for the filtered three-dimensional velocity vector, $\overline{p}$ is the filtered pressure scalar field, ν represents the kinematic viscosity and ρ for the density of the fluid. τ corresponds to the sub grid-scale (SGS) stress tensor:

$$\tau = -2\nu_{SGS}\overline{S} + (\tau : I)I/3 \quad (3)$$

where ν_{SGS} is the sub grid viscosity, $\overline{S}$ is the filtered rate of strain tensor and I the identity tensor.

To close the formulation a suitable diffusive model is needed. Turbulence modeling is carried out in the present paper by using the SIGMA SGS model (Nicoud et al., 2011) in conjunction with wall models within a classical LES framework. The SIGMA model uses the eigenvalues from a matrix derived using the velocity gradient tensor to calculate the turbulent viscosity. The default filter length chosen is twice the cubic root of the cell volume, $l = 2 \times \sqrt[3]{V}$.

2.2. Wall model

A standard algebraic equilibrium wall model was selected (Werner and Wengle, 1991) to perform an initial assessment of the WMLES strategy ap-

plied to automotive aerodynamic studies. This approach has been successfully used in other types of flows such as the flow over the NASA common research model (Lehmkuhl et al., 2016) or the flow over buildings (Dagneva and Bitsuamlak, 2010), among others.

Werner and Wengle wall model

The algebraic approximation proposed by Werner and Wengle (1991) relies on the assumption that the near-wall velocity profile follows the law-of-the-wall throughout the solid wall region. The algebraic expression is fed with LES velocities corresponding to the first off-wall nodes of the mesh. A linear velocity profile or a logarithmic law are used to determine the wall shear stress depending on whether the off-wall nodes are located in the viscous sublayer or above. Once the wall shear stress is evaluated, it is fed back to the LES computation as an accurate diffusive term for the wall face.

$$u^+ = y^+ \quad y^+ \leq 11.81 \quad (4)$$

$$u^+ = 8.3(y^+)^{1/7} \quad y^+ > 11.81 \quad (5)$$

2.3. Numerical method

Using finite volume techniques, the governing equations are discretized on unstructured collocated meshes by means of a second-order conservative scheme (Trias et al., 2014). Such schemes preserve the symmetry inherent in the continuous differential operators. As a consequence, the symmetry preserving schemes ensure the conservation of the kinetic energy balance for flows, and therefore stability, at any Reynolds numbers and using coarse grids. The velocity-pressure coupling is solved by using a fractional-step algorithm, the temporal discretization for both the convective and diffusive

operators is carried out using a two-step linear explicit scheme (Trias et al., 2014), whereas an implicit first-order scheme is implemented for the pressure gradient term. The Poisson equation resulting from the enforcement of the mass conservation is solved using a conjugate gradient solver. The present numerical code and discretization approach has shown high accuracy in different cases involving flows with massive separations (Rodríguez et al., 2011; Lehmkuhl et al., 2013; Rodríguez et al., 2013; Lehmkuhl et al., 2014; Aljure et al., 2014b, 2015). Further information regarding the numerical method can be found in Jofre et al. (2014); Trias et al. (2014); Aljure et al. (2015).

All the CFD simulations are performed using the TermoFluids high performance parallel code (www.termofluids.com). The METIS (Karypis and Kumar, 1998) software is used for domain partitioning while core to core communications are performed by means of the Message Passing Interface (MPI) libraries. Moreover, all simulations are computed on the in-house JFF cluster and FinisTerra II supercomputer at CESGA. Technical information on the JFF cluster and FinisTerra II supercomputer can be found in www.cttc.upc.edu and www.cesga.es, respectively.

Computational time inverted for each simulation is proportional to the number of time-steps solved and their individual cost. For instance, for the largest mesh used, 79×10^6 elements partitioned on 1024 cpus, the required time per iteration is approximately 1.1 s on the FinisTerra supercomputer. About 8.88×10^5 time steps are performed to achieve the final results, totaling a simulation time of approximately 2.78×10^5 CPU hours. This simulation time is approximately 11 days, which is rather large in comparison to steady state approaches such as RANS, which might take several hours on a smaller

cluster.

2.4. Case definition

2.4.1. Geometry and Computational Domain

The geometry to be considered is the fastback DrivAer car model shown in Fig. 1a.

This geometry is placed in a rectangular computational domain modeling a 3/4 open wind tunnel of dimensions $8.8l_{ref} \times 1.75l_{ref} \times 1.3l_{ref}$, where the x-axis corresponds to the stream wise direction, y-axis marks the vertical direction and the z-axis indicates the span wise direction, as shown in Fig. 1b. l_{ref} is the longitude of the car ($4.6m$). The front of the car is located at $2.0l_{ref}$ downwind from the inlet boundary. The outlet boundary is at a distance of $5.8l_{ref}$, measured from the rear end of the body. As can be observed in Fig. 1, the present simulation uses simplified wheels instead of the full detailed geometry used by Heft et al. (2012). This approach has been taken to reduce the computational effort in solving the wheel area over other strategies such as MRF or sliding meshes. This wheel modeling strategy is expected to introduce some error in the results, particularly in the force coefficients, as removing the wheel spokes will certainly influence pressure distribution around the car. Nonetheless, from a more global point of view, the turbulent structures stemming from rotating wheels are some of the most important in automotive aerodynamics, thus, the decision to them in the present work.

Figure 1: (a) Model dimensions. (b)Computational domain - Units in meters.

2.4.2. Boundary Conditions

A constant velocity profile $\mathbf{u} = (u_{ref}, 0, 0)$ is imposed in both the inlet and ground boundaries. The outlet of the domain is modeled using a pressure based boundary condition. As the computational domain is a 3/4 open wind tunnel, symmetry conditions are imposed in the top, front and back

boundaries (see Fig. 1b). No-slip (or the wall model shear stress) conditions are set for the car surfaces and a surface velocity $\omega \times r$, where ω stands for tangential velocity and r the vector from the axis of rotation to the surface, is set for the wheels assuring the tangential velocity is u_{ref} . As was pointed out in the introduction, Reynolds number $Re = u_{ref}L_{ref}/\nu$ independence of the results has been identified by two different authors at different values. Heft et al. (2012) pointed to $Re = 4.87 \times 10^6$ as the value where results are independent, whereas Strangfeld et al. (2013) reported the lower value $Re = 2 \times 10^6$. The Re number most used in different works of $Re = 4.87 \times 10^6$, based on car length, has been reduced to $Re = 2.43 \times 10^6$ to decrease the computational effort associated with the solution of the boundary layer. This lower Re number was also investigated by Heft et al. (2012). Lowering the Re number to simplify boundary layer resolution has been successfully used by other authors in car-like geometries (Krajnović and Davidson, 2005).

2.5. Mesh description

Four meshes have been used, containing approximately 900k (Mesh 1), 6.5M (Mesh 2), 53M (Mesh 3) and 79M (Mesh 4) control volumes. The meshes are built using prism elements in the near wall area. Furthermore, the near wake zone and the zone trailing the mirrors must be refined to correctly predict the flow structures present in these areas and a refinement area is placed around the rotating wheels to correctly predict the flow resulting from this part. The present simulations are focused on the surroundings of the body and the near wake. Simulations resolving the full wake of the cars require much larger meshes which fall out of the scope of the present study.

Fig. 2 shows selected mesh cuts for Mesh 3 (59M), used in the present

(a) Midplane $y=0$

(b) Stream wise $x=1.7$

(c) Stream wise $x=3.5m$

Figure 2: Visualization of cut planes of the mesh at different positions.

investigation for the WMLES analysis. Fig. 2a shows the midplane mesh cut of the car, whereas Figs. 2b-c show a stream wise plane cut at the height of the mirrors and rear wheels, respectively.

2.6. Time integration study

To obtain the numerical results presented in this section, simulations are advanced in time until a statistical stationary flow is reached. Then, in order to ensure converged statistics, instantaneous data is integrated over

(a) Drag coefficient

(b) Lift coefficient

Figure 3: Force coefficients time evolution for Mesh 4

a sufficiently long-time period. To asses the temporal evolution of the flow, several parameters are studied: Time evolution of the force coefficients (Figs. 3 and 4) and non-dimensional pressure coefficient ($c_p = (P - P_\infty)/(0.5\rho U_{ref}^2)$) at selected location around the car, presented in Fig. 5. Note the origin of coordinates is positioned on the floor, in front of the car (see Fig. 1a).

As the flow around the DrivAer model does not reach a steady state, both, drag and lift coefficient (specially the lift coefficient), show an unsteady

(a) Drag coefficient

(b) Lift coefficient

Figure 4: Average evolution depending on start time of integration

behavior. These values fluctuate around a statistical stationary value. Thus, as stated before, to obtain averaged force coefficients that show temporal independence, forces have to be integrated over a time period.

To determine when the initial transient behavior has been washed out and results integrated, a receding average function, similar to that used by Gaylard et al. (2017) was applied. The present a-posteriori analysis consisted in advancing in time and calculating the averaged forces for the specified

Probe	x/l_{ref}	y/l_{ref}	z/l_{ref}
P1	0.38	0.27	0
P2	0.38	0.27	0.20
P3	0.98	0.24	0
P4	0.98	0.24	0.20
P5	1.01	0.13	0
P6	0.87	0.26	0
P7	0.91	0.11	0.18
P8	0.87	0.26	0.13

Figure 5: Numerical probes' location

integration span. Results are plotted on Fig. 4. For the drag coefficient the initial transient behavior washes out around $TU \approx 1.5$, however this is not so straight forward for the lift coefficient. These results evidence low frequency

Figure 6: Pressure coefficient vs time for the different probes

Figure 7: Caption on page 22

phenomena might be influencing the results a longer simulation time is required to correctly capture this parameter. Nonetheless, integration was chosen to start at $TU \approx 2$ ($TU = tu_{ref}/l_{ref}$).

In order to determine a proper time integration period, the pressure coefficient at the different numerical stations is evaluated. Fig. 6 shows the time evolution of this magnitude, where large fluctuations are observed at all locations, including what seem to be an oscillatory or periodic-like behavior.

Figure 7: cont: Autocorrelation function for the pressure time history at the monitored stations

The autocorrelation function was calculated and plotted for the different stations to better understand the unsteady and periodic-like behavior present (see Fig. 7). The signal, for all stations, descends rapidly to the 0 value and then oscillates in time. As observed in Fig. 7, the signals show some oscillatory behavior and different frequencies are observed among the different stations. The largest period observed occurs in stations P4 and P5 (Figs. 7d-e), where the period is around $T = 2.6TU$. Instantaneous data has been

averaged around $TU = 5.2$ time units, capturing approximately 2 cycles of the lowest frequency behavior observed. Larger integration period would be necessary to better understand the effects these low-frequency phenomena exert on the overall results, however, computational limitations have made larger integration times unfeasible. Thus, the lower frequency components will not be included in the present paper.

2.7. Mesh sensitivity study

One of the largest challenges when attempting any LES simulation is the mesh resolution assessment. In the present paper four meshes were constructed, each one relying on information extracted from the previous one. A posteriori analysis of the non-dimensional wall normal distance, in wall units, (y^+) was carried out for each mesh. This distance can be approximated as $y^+ = u_\tau y / \nu$, where u_τ is the friction velocity defined as $u_\tau = \sqrt{|\tau_w|/\rho}$ and τ_w is the wall shear stress, calculated as $\tau_w = \partial u / \partial y$. Average y^+ for the different meshes used are as follows: $y^+ = 272$, $y^+ = 37.2$, $y^+ = 11.8$ and $y^+ = 4.3$ for the meshes 1, 2, 3 and 4, respectively. Furthermore, Fig. 8 shows the y^+ distribution for the first node adjacent to the wall in the four meshes. It should be borne in mind that the approximate locations for the viscous sub layer is defined as $y^+ < 5$, whereas the inner zone and buffer layers are defined for $5 < y^+ < 10$ and $10 < y^+ < 30$, respectively. Thus, Mesh 3 presents a surface resolution that captures only the phenomena within the buffer layer, whereas the surface resolution for Mesh 4 captures some of the phenomena present in the viscous sub-layer and inner zone.

In order to assess the influence of mesh resolution within the simulations, force coefficients are compared to literature available results. In this sense,

Figure 8: y^+ distribution for the different meshes. From top to bottom: Mesh 1, Mesh 2, Mesh 3 and Mesh 4.

the time-averaged drag (C_D) and lift (C_L) coefficients are summarized in Table 1. Also included in the table is the root-mean-square for the lift coefficient C_{LRMS} . Alongside the numerical results, a brief description of the case set-

ups used for this analysis is included. For more convenience, a simple system will be used to describe the different case set-ups. The acronym M3_S_WM describes Mesh 3 (M3) using the SIGMA SGS model (S) and the wall model (WM). The first part of the acronym describes the mesh used (M1:mesh 1, M2: mesh 2, M3: mesh 3 or M4:mesh 4), the second part indicates the SGS model used (S:SIGMA, N: no model) and the final part references to the use of the wall model (WM: wall model used, NW: no wall model used).

In general, force coefficient results present some scattering, i. e., drag

Case	Mesh	SGS	WM	C_D	C_L	C'_L	C_{LRMS}
M1_S_NW	1	SIGMA	NO	0.293	0.036	-0.015	0.023
M2_S_NW	2	SIGMA	NO	0.253	0.057	-0.006	9.6
M3_S_NW	3	SIGMA	NO	0.246	0.083	0.018	0.024
M4_S_NW	4	SIGMA	NO	0.251	0.069	-0.006	0.021
Exp. ¹				0.265		-0.06	-
EARSM ²				0.2254		0.0322	-
DES ³				0.266		0.024	-
$k - \omega$ SST ⁴				0.2599		-	-
Exp(no GS) ⁵				0.261		0.01	-
DES SST(no GS) ⁶				0.2662		0.0235	-
SST-IDDES(no GS) ⁷				0.268		0.011	-

Table 1: Force coefficients and comparison to selected numerical and experimental results.

¹: Heft (2014), ²: Guilmineau (2014b), ³ Peters et al. (2015), ⁴: Shinde et al. (2013), ⁵: Strangfeld et al. (2013), ⁶: Guilmineau (2014a), ⁷: Ashton et al. (2016)

coefficient results vary between $C_D = 0.2254$ (Guilmineau, 2014b) to $C_D = 0.268$ (Ashton et al., 2016). These differences can be attributed to two main reasons: Low frequency behavior of some quantities requiring a longer integration time, and differences in experimental set up and CFD. In fact, as stated by Hetherington and Sims-Williams (2004, 2006) model mounting strategies (i.e. roof stinger), such as those used by Heft et al. (2012) have a combined effect on drag and lift forces, and wake topology. Furthermore, reported data for the lift coefficient does not include the forces exerted by the wheels, this is shown in table 3 as CL'. It should be noted that there is a large difference in this quantity between experiments and simulations, and there exists some uncertainty on the lift coefficient results obtained in the present investigation.

Finally, the Re number influence on these coefficients should be considered, as well as the rotating simplified wheels, as the resulting wake and pressure distribution might differ from that found in the experiments.

The obtained drag results fall the range of literature available results, however other approaches seem to predict a value closer to experimental data. The obtained results for the lift coefficient present important discrepancies. Results converge towards the obtained drag coefficient of $C_D = 0.251$ and lift coefficient of $C_L = 0.07$.

A second parameter studied to assess mesh resolution were averaged pressure coefficient profiles. Three profiles are plotted in Fig. 9: along the top and bottom of the car's mid plane and on the horizontal $y = 0.03l_{ref}$ plane. Similarly as with the force coefficients, these profiles are compared to the pressure measurements reported by Heft et al. (2012) and numerical results

(a) Top

(b) Bottom

Figure 9: Caption on page 28

from Guilmineau (2014a) and Ashton et al. (2016). Small discrepancies are found through the distribution profile, however, the largest is found around $x/l_{ref} = 0.45$. As pointed by Peters et al. (2015) and Pasquali et al. (2015),

(c) $y=0.03l_{ref}$

Figure 9: cont: Average pressure coefficient profile over (a) the top of the car geometry in the mid plane, (b) the bottom of the car geometry in the mid plane and (c) the side of the car geometry in the horizontal $y=0.03l_{ref}$ plane

at this location the roof stinger is set up in the wind tunnel experiments, which is not placed in the CFD simulation. In addition, Peters et al. (2015) also pointed out that there are some discrepancies between the CAD and the experimental model in the windshield-roof transition. Results for the three profiles converge to the same curves for the larger meshes, which indicates grid resolution is enough to capture the flow phenomena present around the car.

Observing the force coefficients in Table 1, mesh independence of the forces is reached at Mesh 3. Note that for Case 2, although force coefficients are well within the expected values, lift RMS is very high. This indicates the mesh is not fine enough to correctly solve the flow field. Pressure profiles converge towards the results observed in Mesh 4. The surface mesh refinement

carried out for mesh 4 decreases the mean y^+ value allowing the boundary layer's inner zone to be better resolved.

3. RESULTS

3.1. Time step and CPU time

One of the main concerns when using LES simulations in industry applications is the large computational expense required. As meshes are finer a smaller time step is required in the simulations. In addition, the larger meshes require more time (or CPUs) to solve. Table 2 shows the non-dimensional time step, iteration time, number of iterations and total CPU time required for the simulations of cases M3_S_WM and M4_S_NW.

As can be seen in table 2, there is a nearly 60% reduction in the time step size when changing from Mesh 3 to Mesh 4 due to the smaller control volumes around the car's surface. This time step decrease causes an increase of 130% in the number of iterations required to simulate the time span required in the present case. Using the WM considerably reduces the computational requirements needed for the present simulations, CPU time is reduced in approximately 70%.

Case	dt [TU]	iter	t_{iter} [s]	t_{cpu} [CPU hours]
M3_S_WM	2.03×10^{-5}	3.86×10^5	$0.7s$	8.23×10^4
M4_S_NW	8.83×10^{-6}	8.88×10^5	$1.1s$	2.78×10^5

Table 2: Average time step dt , iteration time t_{iter} , number of iterations and overall time t_{cpu} required for the different numerical configurations.

3.2. WMLES analysis

One of the main objectives of the present paper is to assess the validity of flow solutions using WMLES. Additionally, a case was simulated with no SGS modeling, also presented here for comparison.

Table 3 shows the different set-ups studied and the resulting time averaged force coefficients. The use of the wall model in mesh three reduces the difference present in the drag coefficient between Mesh 3 and Mesh 4, for both SGS models. Nonetheless, there seems to be an improvement in the results when using the wall model when compared to the finer mesh results.

A more detailed look is taken to examine the effects that the wall model exerts on the flow around the car. Pressure profiles are analyzed to assess small scale changes present when using the wall model. Results using Mesh 3 are compared to those obtained with Mesh 4, both using the SIGMA model (with and with out WM). The case M3_N_NW is included for reference.

In Fig. 10, the pressure profiles in the mid plane along the top of the car are plotted. Cases M3_S_NW, M3_S_WM and M3_N_NW are compared

Case	Mesh	SGS	WM	C_D	C_L	C_{LRMS}
M3_S_NW	3	SIGMA	NO	0.246	0.083	0.024
M3_S_WM	3	SIGMA	YES	0.251	0.056	0.024
M4_S_NW	4	SIGMA	NO	0.251	0.069	0.021
M3_N_NW	3	NO SGS	NO	0.252	0.064	0.022

Table 3: Force coefficients for the different numerical configurations and experimental results.

(a) Top

(b) Bonnet-Wind shield (BW-J)

(c) Fastback (FB)

Figure 10: Averaged pressure coefficient along the top of the car in the mid plane.

to case case M4_S_NW (finer mesh). When using WM almost no changes are observable for most of the cars length, except for two zones where the effects are more notorious: the bonnet-windshield junction (BW-J) and the fastback (FB) (see Figs. 10b-c).

Fig. 11 shows the averaged streamlines around the car and close ups to the two section of interest identified previously. The BW-J has a small corner

(a) Top

(b) Bonnet-Wind shield junction

(c) Fastback

Figure 11: Streamlines in the mid-plane of the car, colored by non dimensional pressure coefficient

that greatly affects the flow configuration. As the flow moves downstream over the bonnet, the gradual pressure increase it experiences is suddenly cut as the junction is reached and flow tends to separate (see section 3.3). This separation creates a small recirculation zone, shown in Fig. 11b. As observed in Fig. 10b, the influence of the wall model in this area affects

the results negatively. In the front part of the junction, where the flow detaches, the wall model yields slightly closer results to the finer mesh, however, as we move into the junction the pressure decrease predicted by the SIGMA+WM set-up is larger than the SIGMA model alone. Both pressure drops are larger than that predicted by the finer mesh. In the final part of the junction the SIGMA+WM set-up shows a larger pressure recuperation than that showed by the SIGMA model alone. The pressure recuperation resulting from the finer mesh is smaller. Finally, at this area, the results yielded by case M3_N_NW (no turbulence modeling) are quite different than the rest.

Results in the FB area behave differently. Observing Fig. 10c, two areas of interest are observed. The first one, around $x/l_{ref} \approx 0.8$, corresponds to a small lip located at the junction between the roof and the FB. In this area, similarly as was observed in the BW-J region, flow detaches, generates a recirculation area and a low pressure spike appears (Fig. 11c). Results obtained for case M3_S_WM do not capture well this change in pressure, over predicting the drop. Results for Cases M3_S_NW and M4_S_NW are nearly identical. Again, results from case M3_N_NW are very different than the rest.

As the flow moves along the FB a second zone of interest is found. Between $0.75 \leq x/l_{ref} \leq 1$ the pressure results obtained show differences for the different case set ups. Observing Fig. 11c, several small recirculation bubbles are observed. The formation of these structures is somewhat different to the separation regions observed for the BW-J or the roof-rear window lip. Flow separation is not induced by a sudden geometrical discontinuity as is the case

in the edge in the BW-J or the small lip in the junction between the roof and the rear window. In the FB area a gradual change in the geometry causes the flow to separate and reattach several times. These small separation areas do not induce a large pressure drop. On the contrary, there is a pressure recuperation over this area. For this zone, results from case M3_S_WM are closer to those of case M4_S_NW than the others, indicating a positive effect of using the wall model in this area (Fig. 10c).

Fig. 12 shows the pressure profiles along the mid plane at the bottom of the car. As has been pointed out by diverse authors (Heft et al., 2012; Guilmineau, 2014b, among others), the pressure coefficient is negative over the whole under body (positive values in the distribution correspond to the small area between the stagnation point and the under body), acting as down force generator. Again, minor differences are observed in these profiles when using the wall model, although two areas stand out: the front of the under body near the wheels (U-F), shown in Fig. 12b and the diffuser area (D), in Fig. 12c. For the U-F area, the pressure prediction for cases M3_S_NW and M3_S_WM are different, being those from case M3_S_NW a bit closer to case M4_S_NW. Observing Fig 11a, this area coincides with the front edge of the under body. Flow separates in this region generating a significant pressure drop, seen in Fig. 12b for $x/l_{ref} \approx 0$. The subsequent area is the pressure recovery from this separation region, which is over predicted by the wall model.

On the other hand, for area D (Fig. 12) the wall model (case M3_S_WM) yields closer results to the finer mesh than those obtained for case M3_S_NW, showing a positive effect of the wall model. This area is not subject to flow

separation and the diffuser increases the flow area under the car permitting the pressure recovery to be done gradually. A side effect of the pressure recovery experienced in the diffuser is the low pressure spike observed at around $x/l_{ref} \approx 0.7$. This pressure spike is responsible for a large part of the downforce generated by diffusers in automotive applications. This

(a) Bottom

(b) Front (U-F)

(c) Diffuser (D)

Figure 12: Averaged pressure coefficient along the bottom of the car in the mid plane.

pressure decrease is well captured by the wall model, being results from case M3_S_WM closer to case M4_S_NW than those predicted by case M3_S_NW. Similarly as with the pressure profiles over the top, when using no turbulence model results drift away from those obtained by case M4_S_NW. This is specially evident the diffuser.

The pressure distribution along the side of the car shows a different behavior (see Fig. 13). This pressure profile is obtained at the $y=0.03l_{ref}$ plane, at the height where the front stagnation point is located. Flow around this area is heavily influenced by the unsteady effects brought on by the wheel's rotation. Results in the wheel well area, between $0.1 \leq x/l_{ref} \leq 0.25$ for the front and $0.7 \leq x/l_{ref} \leq 0.85$ for the rear, show small changes when using the wall model. Nonetheless, results do not improve when using the wall model. In the areas just after the wheel wells a pressure drop is observed. This pressure drop is generated by the corner present in the car's wheel well.

Figure 13: Averaged pressure coefficient over the side of the car geometry in the horizontal $y=0.03l_{ref}$ plane

This pressure decrease is well captured by the wall model, however, the pressure recuperation is under predicted at the front and over predicted at the back, when compared to the finer mesh results.

The different behavior the WM shows in the regions of the car is strongly influenced by the flow phenomena occurring. As the Werner and Wengle WM was created for equilibrium flows that show a log-law behavior, heavy, geometrically induced, separation regions which result in large recirculation areas fall out of its scope of applicability. This is the case of the BW-J or the U-F areas. On the other hand, the gradual geometry changes such as those found in the FB and D regions, allow a boundary layer development which might resemble more that found in a developed boundary layer along a flat plate.

In order to fully assess the effect the wall model exerts on the flow configuration, results obtained in different spanwise locations are here compared. As has been observed, one of the areas were the wall model works best is in

Figure 14: Span wise locations for the pressure profiles at the fastback section.

(a) $z/l_{ref} = 0$

(b) $z/l_{ref} = 0.05$

Figure 15: Caption on page 39

the FB region. In the present section, the pressure profiles along the FB and at the different span wise locations $z/l_{ref} = 0$, $z/l_{ref} = 0.05$, $z/l_{ref} = 0.1$ and $z/l_{ref} = 0.125$ (see Fig. 14) are studied and plotted on Fig. 15. For comparison sake, the pressure profiles reported by Guilmineau (2014a) and Strangfeld et al. (2013) are plotted as well. It should be noted that the results by Strangfeld et al. (2013) are uncorrected and the closed wind tunnel may have had caused an offset in the measurements.

(c) $z/l_{ref} = 0.1$

(d) $z/l_{ref} = 0.125$

Figure 15: cont: Average pressure coefficient profile over the fastback at different span wise locations and using different SGS models. (a) $z/l_{ref} = 0.00$, (b) $z/l_{ref} = 0.05$, (c) $z/l_{ref} = 0.1$, (d) $z/l_{ref} = 0.125$. Comparison with Guilmineau (2014a)

Three regions stand out in Fig. 15, the lip in the junction between the roof and the FB ($x/l_{ref} \approx 0.77$), the FB region ($0.8 \leq x/l_{ref} \leq 0.95$) and the rear end of the car ($x/l_{ref} \geq 0.95$). Again, the wall model seems to produce mixed results.

For the span wise locations $z/l_{ref} = 0$ and $z/l_{ref} = 0.05$ (Figs. 15a-b)

the wall model has the same negative effect on the solution, over predicting the pressure drop in the separation region. As the flow moves over the FB, the initial pressure recovery from the separation in the roof-rear window transition shows the same difficulties observed in the previous section. Small differences are observed in most of the FB area, particularly around $x/l_{ref} \approx 0.9$, the zone related with the different recirculation bubbles present along the rear window (see Fig. 11c). Nonetheless, the effect from the wall model is similar, having an overall closer prediction to the finer mesh of the pressure increase over the rear window than its non wall model counter part. At the rear-most section of the FB small differences are observed between cases M3_S_NW and M3_S_WM. In this area the WM does not seem to improve the results. Note the small pressure spike present at around $x/l_{ref} \approx 0.97$, where results obtained from case M3_S_NW are closer to the finer mesh than those obtained using the wall model.

For the span wise location $z/l_{ref} = 0.1$ (Fig. 15c), pressure profiles show less discrepancies between the different case set-ups, particularly in the spoiler and FB section. Nonetheless, some improvement is observed in the rear section when using the WM. Again, the wall model seems to improve the overall pressure prediction in this area. Finally, for the outermost location, $z/l_{ref} = 0.125$ (Fig. 15d), the results show the WM do not improve substantially the results. At this span wise location, there is no spoiler, but in the same stream wise location the profile crosses over the c-pillar. The wall model seems to improve on the results at the c-pillar. On the contrary, results using the WM are farther away from the finer mesh at the rear of the car.

3.3. Flow Structures

Flow around the DrivAer model is quite complex, where plenty of vortices are formed around the geometry. To help the reader follow the contents of this section, Fig. 16 show the nomenclature used for different parts of the car. Unsteady flow is analyzed by extracting the coherent structures around the car, obtained from case *M3_S_WM*. To do this non-dimensional stream wise vorticity $\omega^* = \omega l_{ref}/U_{ref}$ and non-dimensional Q-criterion $Q^* = Q l_{ref}^2/U_{ref}^2$ (Hunt et al., 1988) contours for case *M3_S_WM* in 80 different time instants are studied. Representative snapshots are presented in the figures in the present section. These figures depict the $Q^* = 1$ and $\omega^* \pm 50$ iso contour surfaces viewed from the front and the back of the car, respectively. Additionally, the average pressure coefficient over the car's surface and average $Q_{avg}^* = 0.2$ is also presented in Figs. 18 and 21.

As was pointed out in section 3.2, several locations around the car present flow separation and recirculating regions. As these zones are located all over the car, its resulting wake presents a very strong unsteady behavior.

When observed from the front, several regions stand out (Fig. 17 and 18). As the flow impacts the front of the car, it travels through the top part of the bonnet. A stagnation zone is created in the front bumper area and a high pressure bubble encloses most of the car's frontal section (see Fig. 18a). Flow then travels upwards towards the bonnet and side wards towards the wheel well area, detaching from the front edge of the bonnet.

In the top of the bonnet, the flow experiences an adverse pressure gradient causing it to decelerate and it presents the streak instability behavior. This feature consists of small lifted structures from the car's surface, marked as

Figure 16: DrivAer model and the nomenclature used for its different parts

1 in Fig. 17b. As the flow reaches the BW-J, the geometrical discontinuity present there causes the flow to separate again and to generate a recirculation region. As a consequence, the flow detaches from the edge, generating small horseshoe-like vortices that travel up the windshield (see region 1 in Fig. 17a).

In this area a secondary high pressure bubble is created (see Fig. 18a), indicating the reattachment of the flow to the windshield surface. As the flow reaches the top of the windshield it detaches again, creating small scale structures (see Fig. 18b). In addition, as the horseshoe-like vortices reach the top edge of the windshield they merge with the small structures present there, creating larger longitudinal vortex that travel over the top of the car (see region 2 in Fig. 17a). This process takes place at different span wise locations feeding instability to the flow downstream.

Along the lateral of the bonnet the flow behaves differently as it does on the its top. As the flow moves over the top corner of the front it detaches, in

(a) $Q^* = 1$

(b) $\omega^* \pm 50$

Figure 17: Instantaneous turbulent structures around the car viewed from the front, colored by velocity magnitude. (a) $Q^* = 1$ iso surfaces. (b) non-dimensional stream wise vorticity $\omega^* \pm 50$.

a similar manner as it does on the center of the bonnet, however due to the influence of the bonnets side edge, the flow reattaches. Reattachment zones are accompanied by a rise in the surface pressure. The resulting pressure distribution along the surface in the side of the car's nose redirects part of

(a) C_p

(b) $Q_{avg}^* = 0.2$

Figure 18: (a) Averaged pressure coefficient C_p over the surface of the car. (b) averaged non-dimensional $Q_{avg}^* = 0.2$ contours

the flow downwards, under the mirror (see region 2 in Fig. 17b).

The secondary high pressure zone created in the windshield pushes the flow outwards, towards the A-pillar. Interactions around this area create vortical structures that move downstream forming the A-pillar vortex (see region 3 in Fig. 17a). This structure travels along the top corner of the car

and join the flow in the FB region. Furthermore, the mirrors create a large unsteady wake that travels alongside the car and joins the main wake behind the car (see region 4 in Fig. 17a). Both these large scale structures, also observed in Fig. 18b, are noise sources in external car aerodynamics.

In order to better understand this unsteady phenomena, a fast Fourier transform was performed on the pressure values for the different numerical probes located around the car (Fig. 5), yielding the frequency spectra at the specified locations. Frequency is non-dimensionalized with the vehicle's base height ($h_{base} = 0.72m$) and reference velocity $St = fh_{base}/U_{ref}$. It should be noted that low frequency behavior is not well captured in the present simulations.

Numerical probe P1 is located on top of the windshield area, whereas probe P2 is located over the A-pillar.

Fig. 19 shows the frequency spectra obtained for probes P1 and P2. A high frequency broad band peak is observed around $St \approx 6.2$ for station

Figure 19: Energy spectra for the pressure fluctuations at stations P1 and P2.

P1. The location of this probe suggests this corresponds to the small scale vortex activity around the top of the windshield. Additionally, a narrower, more energetic peak is also visible around the non-dimensional frequency $St \approx 1.2$. As Wieser et al. (2013) stated, frequencies around this range correspond to shedding phenomena around the car. With this in mind, this energy peak might correspond to the shedding of large scale vortices from the top of the windshield. A similar peak is observed around $St \approx 1.45$ for station P2, corresponding to the A-pillar vortex shedding. The larger energy peak, observed at station P2 at $St \approx 0.17$ corresponds to the vortices forming at the mirror.

Other structures which are visible from the front correspond to the wheels (region 5 in Fig. 17). Both, front and rear wheels, interact heavily with the oncoming flow, creating large scale structures alongside the car. It should be noted that the jetting vortices created by the wheels are the largest structures around the car.

When viewed from behind, the large vortices created by the A-pillar, mirror and wheels are also visible (see Figs. 20 and 21). In fact, these vortices and their interactions with the car downstream, form most of the large trailing vortices that dominate the wake. However, vortices created in the back of the car play also an important role in the wake's formation.

Flow traveling through the car's roof reaches the small lip junction between the roof and the rear window. As was observed in section 3.2, this region presents flow recirculation and flow detachment forming small scale structures (marked in Fig. 20b as region 1). As it can be observed in Fig. 21a, the whole FB is subject to an adverse pressure gradient, which con-

(a) $Q^* = 1$

(b) $\omega^* \pm 50$

Figure 20: Turbulent structures around the car, viewed from the back, colored by velocity magnitude. (a) $Q^* = 1$ iso surfaces. (b) non-dimensional stream wise vorticity $\omega^* \pm 50$.

tribute to form small longitudinal vortices along the FB (see region 1 in 20a). These structures travel downwind and detach from the back of the car.

(a) C_p

(b) $Q_{avg}^* = 0.2$

Figure 21: (a) Averaged pressure coefficient C_p over the surface of the car. (b) averaged non-dimensional $Q_{avg}^* = 0.2$ contours

As it was mentioned before, the A-pillar vortices reattach to the car in the FB region, enclosing the flow in this area. Evidence of the reattachment can be seen as two localized high pressure zones in the vehicles rear end, see Fig. 21. Finally, the C-pillar vortices are also visible in Fig. 20, marked as region 2. The present geometry shows a very small C-pillar vortex, barely visible in Fig. 20 and not visible in 21, when compared to other simplified fast-back

Figure 22: Energy spectra for the pressure fluctuations at stations P3, P4, P6 and P8.

cars, such as the 25° slant back Ahmed model (Aljure et al., 2014b).

The different numerical probes located at the model's rear section aim to capture the different unsteady phenomena taking place. Fig. 22 depicts the frequency spectra obtained for probes P3, P4, P6 and P8. Probe P3 is located in the center of the car's rear end, P4 is located in the car's rear end corner, P6 in the center of the rear window and P8 over the C-pillar. In rear center of the car (P3 and P6), high frequency broad band peaks are

Figure 23: Frequency spectra on the DrivAer model at the locations specified on Fig. 5.

observed around $St \approx 2.5$. These correspond, in a similar way as for P1, to the shedding of small scale vortices (Figs. 22a,c).

A lower frequency peak is observed for stations P4 and P8 around $St \approx 1$. Similarly as with P1 and P2, these peaks correspond to the shedding of large structures around the car. In the case of P8 the energy peak observed around $St \approx 1.05$ for Fig. 22d is, by far, the most energetic phenomena observed. Its location suggests it identifies the large trailing vortices that dominate the wake which are formed by different structures forming around the car. The large vortex generated in the mirror is captured by the probe located at station P4. This structure shows a very energetic peak at the frequency $St \approx 0.15$, similar to that observed at station P2 at $St \approx 0.17$.

Figs. 23a-b, show the frequency spectra for the numerical probes located at the rear bumper and wheel wake areas (P5 and P7). As was described by Duell and George (1999), there is a pumping effect present in the near wake of car-like geometries. This effect has also been experimentally observed by

Wieser et al. (2013) for the DrivAer model. Both, Duell and George (1999) and Wieser et al. (2013) report a pumping frequency around $St = 0.07$. Station P5 is located in this area (Fig. 23a). Energy spectra for the pressure fluctuations in this region show a frequency of $St = 0.08$, indicating this pumping effect seems to be captured in the present simulations, however, the large period of this phenomena requires a longer simulation time to confirm. Finally, a high peak at $St = 0.27$, close to the wheel rotation frequency ($St \approx 0.3$) is seen for station P7 (Fig. 23b). This energy peak is probably associated to the jetting vortices created by the interaction of the wheels and the ground (Aljure et al., 2014a).

4. CONCLUSIONS

In this work LES and WMLES simulations of the flow around the DrivAer car model have been carried out using smooth under body, mirrors, simplified wheels and moving ground. In order to reduce the computational power required to solve the boundary layer-related phenomenology, a reduced Reynolds number and a simplified wheel geometry has been used. Two values have been reported regarding Reynolds number independence on this geometry. Heft et al. (2012) reported results are independent of Reynolds number after $Re = 4.87 \times 10^6$, whereas Strangfeld et al. (2013) reported the lower $Re = 2 \times 10^6$ value. Taking this into account, the present simulations were ran using $Re = 2.43 \times 10^6$. Both of these factor influence obtained values and some differences between literature results and present ones is expected. Simulations were ran using 4 meshes, containing 900k (Mesh 1), 6.5M (Mesh 2), 53M (Mesh 3) and 79M (Mesh 4) control volumes each. Furthermore, the

Werner and Wengle wall model was implemented on Mesh 3, in conjunction with the SIGMA turbulence model, and results compared to those yielded by the finer mesh. Finally, a study into the turbulent behavior and unsteady phenomena around the car has been performed.

The use of WM allows simulations to be carried out in coarser meshes, particularly in the near wall regions. The larger elements in these areas allows a larger time step to be used, and thus, a lower number of iterations. In the present case, CPU time required for Mesh 3 with the wall model is around 70% lower than that required for Mesh 4.

The very complex flow present around this geometry result in some scattering of the literature available results. The drag coefficient results are well converged, however a large difference is found with experimental values. Regarding the lift coefficient, low frequency phenomena seems to be influencing the results and longer simulations times might be necessary. A drag coefficient of $C_D = 0.251$ and a lift coefficient of $C_L = 0.07$ are obtained in the present simulations. Furthermore, as the computational power is focused around the wall regions prioritizing it over the wake area, the latter might be under resolved, tainting the results. Although there are discrepancies with the reported results, present simulations are useful to evaluate the WMLES potential.

Force coefficients show improvement when using the WM. Using the SIGMA turbulence model drag coefficient changes from $C_D = 0.246$ without WM to $C_D = 0.251$ using the WM on Mesh 3, compared to the value of $C_D = 0.251$ obtained with Mesh 4. The change in the lift coefficient is more significant, from $C_L = 0.083$ without WM to 0.056 with WM, compared to

the obtained value of $C_L = 0.069$ in Mesh 4. Nonetheless, results for the lift coefficient are inconclusive and further investigation is required to assess this parameter.

Small differences are observed in the pressure profiles, and using the WM gives mixed results. Improvement is observed in zones with adverse pressure gradient and smooth geometrical changes. In the present geometry this zones include the fastback and diffuser areas. Results obtained in zones where the geometry triggers flow separation and recirculation do not show improvement, such as the junction between the bonnet and windshield or the small lip between the roof and rear window. Overall analysis of the models leads to the conclusion that, from a practical viewpoint, the WMLES using the Werner and Wengle wall model improves the obtained results with very low computational cost.

One of the strongest features LES simulation have is the ability to study flow structures and unsteady effects around the car. This is a very useful tool as it can better predict large scale structures that directly affect aerodynamic behavior, as well as, unsteady effects which can have an important impact on ride stability and noise generation. Flow around the DrivAer geometry is quite complex and a lot of interactions create plenty of vortical structures around the car. The most important structures observed are the mirror induced vortices and those created by the rotating wheels. Characteristic frequencies for these phenomena have been measured to be $St \approx 0.15$ for the mirror vortex and $St \approx 0.3$ for the wheel vortices. The shedding of large structures around the car show a non-dimensional frequency of $St \approx 1$. This structures include the A-pillar vortex and vortices detaching of off the

windshield-roof junction. Finally, Smaller scale phenomena show a much larger frequency. Small structures detaching of the BW-J area and those forming in the FB area show non-dimensional frequencies around $St \approx 6.2$ and $St \approx 2.5$, respectively.

ACKNOWLEDGMENTS

This work has been partially financially supported by the “Ministerio de Economía y Competitividad, Secretaría de Estado de Investigación, Desarrollo e Innovación”, Spain (Project: “Development of high performance parallel codes for the optimal design of thermal equipment, reference ENE2010-17801”), by the collaboration project between “Universitat Politècnica de Catalunya” and Termo Fluids S.L. and the “Departamento Administrativo de Ciencia, Tecnología e Innovación - Colciencias” through their doctoral training program “Francisco Jose de Caldas”.

BIBLIOGRAPHY

- Adams, N. A., Hickel, S., 2009. Implicit Large-Eddy Simulation: Theory and application. In: Advances in Turbulence XII, Marburg, Germany.
- Ahmed, S. R., Ramm, G., Faltin, G., 1984. Some salient features of the time averaged ground vehicle wake. SAE paper no 840300.
- Aljure, D. E., Lehmkuhl, O., Martinez, D., Favre, F., Oliva, A., 2014a. On the IBM approximation for the wheel aerodynamic simulation. In: First international conference in numerical and experimental aerodynamics of road vehicles and trains, Bourdeaux, France.

- Aljure, D. E., Lehmkuhl, O., Rodríguez, I., Oliva, A., 2014b. Flow and turbulent structures around simplified car models. *Comp. Fluids* 96, 122–135.
- Aljure, D. E., Rodríguez, I., Lehmkuhl, O., Pérez-Segarra, C. D., Oliva, A., 2015. Influence of rotation on the flow over a cylinder at $Re = 5000$. *Int. J. Heat Fluid Fl.* 55, 76–90.
- Ashton, N., West, A., Lardeau, S., Revell, A., 2016. Assessment of RANS and DES methods for realistic automotive models. *Comput. Fluids* 128, 1–15.
- Bello-Millán, F. J., Mäkelä, T., Parras, L., del Pino, C., Ferrera, C., 2016. Experimental study on ahmed’s body drag coefficient for different yaw angles. *J. Wind Eng. Ind. Aerod.* 34 (6), 1111–1119.
- Bonnavion, G., Cadota, O., Évrard, A., Herbert, V., Parpais, S., Vigneron, R., Déleroy, J., 2017. On multistabilities of real car’s wake. *J. Wind Eng. Ind. Aerod.* 164, 22–33.
- Chen, H., Kandasamy, S., Orszag, S., Shock, R., Succi, S., Yakhot, V., 2003. Extended boltzmann kinetic equation for turbulent flows. *Science* 301 (5633), 633–636.
- Dagnewa, A. K., Bitsuamlak, G. T., 2010. LES evaluation of wind pressures on a standard tall building with and without a neighboring building. In: *The Fifth International Symposium on Computational Wind Engineering (CWE2010)*, Chapel Hill, USA.
- Duell, E., George, A., 1999. Experimental study of a ground vehicle body unsteady near wake. *SAE Technical Paper* (1999-01-0812).

- Gaylard, A. P., Kabanovs, A., Jilesen, J., Kirwan, K., Lockerby, D. A., 2017. Simulation of rear surface contamination for a simple bluff body. *J. Wind Eng. Ind. Aerod.* 165, 13 – 22.
- Gilhome, B. R., 2002. Unsteady flow structures and forces over/on the rear window and boot lid of sedan automobiles. In: *Progress in vehicle aerodynamics III - unsteady flow effects*.
- Guilmineau, E., 2014a. Numerical simulations of flow around a realistic generic car model. *SAE Int. J. Passeng. Cars - Mech. Syst.* 7 (2).
- Guilmineau, E., 2014b. Numerical simulations of ground simulation for a realistic generic car model. In: *Proceedings of the ASME 2014 Joint US-European Fluids Engineering Division Summer*, Chicago, USA.
- Heft, A. I., 2014. Aerodynamic investigation of the cooling requirements of electric vehicles. In: *PhD Thesis, Technical University Munich*. No. ISBN 978-3-8439-1765-0 3. Verlag Dr. Hut.
- Heft, A. I., Indinger, T., Adams, N. A., 2012. Introduction of a new realistic generic car model for aerodynamic investigations. *SAE Technical Paper* (2012-01-0168).
- Hetherington, B., Sims-Williams, D., 2004. Wind tunnel model support strut interference. *SAE Technical Paper* (2004-01-0806).
- Hetherington, B., Sims-Williams, D., 2006. Support strut interference effects on passenger and racing car wind tunnel models. *SAE Technical Paper* (2006-01-0565).

- Hunt, J. C. R., Wray, A. A., Moin, P., 1988. Eddies, stream and convergence zones in turbulent flows. In: Proceedings of the Summer Program 1988.
- Islam, A., Gaylard, A., Thornber, B., 2017. A detailed statistical study of unsteady wake dynamics from automotive bluff bodies. *J. Wind Eng. Ind. Aerod.* 171, 161–177.
- Jakirlic, S., Kutej, L., Hanssmann, D., Basara, B., Schütz, T., Tropea, C., 2014. Rear-end shape influence on the aerodynamic properties of a realistic car model: a RANS and hybrid LES/RANS study. In: *New Results in Numerical and Experimental Fluid Mechanics X*, 9th STAB/DGLR Symposium Munich, Germany. pp. 397–407.
- Jofre, L., Lehmkuhl, O., Ventosa, J., Trias, F. X., Oliva, A., 2014. Conservation properties of unstructured finite-volume mesh schemes for the Navier-Stokes equations. *Numer. Heat Tr. B-Fund.* 64 (1), 53–79.
- Karypis, G., Kumar, V., 1998. Multilevel k-way partitioning scheme for irregular graphs. *J. Parallel Distr. Com.* 48, 96–129.
- Kawai, S., Larsson, J., 2013. Dynamic non-equilibrium wall-modeling for large eddy simulation at high Reynolds number. *Phys. Fluids* 25, 015105.
- Khalighi, B., Jindal, S., Iaccarino, G., 2012. Aerodynamic flow around a sport utility vehicle computational and experimental investigation. *J. Wind Eng. Ind. Aerod* 107, 140–148.
- Krajnović, S., Davidson, L., 2005. Flow around a simplified car: Part1: Large eddy simulation. *ASME: Journal of Fluids Engineering* 127, 907–925.

- Larsson, J., Kawai, S., Bodart, J., Bermejo-Moreno, I., 2016. Large eddy simulation with modeled wall-stress: recent progress and future directions. *Mech. Eng. reviews* 3 (1).
- Le Good, G., Garry, K., 2004. On the use of reference models in automotive aerodynamics. SAE Technical Paper (2004-01-1308).
- Lehmkuhl, O., Park, G. I., Moin, P., 2016. LES of flow over the NASA common research model with near-wall modeling. In: *Proceedings of the summer program 2016, center for turbulence research, Stanford*. pp. 335–341.
- Lehmkuhl, O., Rodríguez, I., Borrell, R., Chiva, J., Oliva, A., 2014. Unsteady forces on a circular cylinder at critical Reynolds numbers. *Phys. of Fluids* 26, 125110.
- Lehmkuhl, O., Rodríguez, I., Borrell, R., Pérez-Segarra, C. D., Oliva, A., 2013. Low-frequency variations in the wake of a circular cylinder at $Re = 3900$. *Phys. of Fluids* 25 (8), 085109.
- Lichtneger, P., Ruck, B., 2017. Full scale experiments on vehicle induced transient pressure loads on roadside walls. *J. Wind Eng. Ind. Aerod.* in Press.
- Nicoud, F., Toda, H., Cabrit, O., Bose, S., , Lee, J., 2011. Using Singular Values to Build a Subgrid-Scale Model for Large Eddy Simulations. *Phys. Fluids* 23.
- Park, G. I., Moin, P., 2014. An improved dynamic non-equilibrium wall-model for large eddy simulation. *Phys. Fluids* 26, 015108.

- Pasquali, A., Schönherr, M., Geier, M., Krafczyk, M., 2015. Simulation of external aerodynamics of the drivAer model with the LBM on GPGPUs. In: *Advances in Parallel Computing, Parallel Computing: On the Road to Exascale*. Vol. 27. pp. 391–400.
- Peters, B. C., Uddin, M., Bain, J., Curley, A., Henry, M., 2015. Simulating drivAer with structured finite difference overset grids. SAE Technical Paper (2015-01-1536).
- Piomelli, U., 2008. Wall-layer models for large-eddy simulations. *Prog. Aerosp. Sci.* 44, 437–446.
- Piomelli, U., Balaras, E., 2002. Wall-layer models for large-eddy simulations. *Annu. Rev. Fluid Mech.* 34, 349–374.
- Rodríguez, I., Borrell, R., Lehmkuhl, O., Pérez-Segarra, C. D., Oliva, A., 2011. Direct numerical simulation of the flow over a sphere at $Re = 3700$. *J. of Fluid Mech.* 25 (25), 263–283.
- Rodríguez, I., Lehmkuhl, O., Borrell, R., Oliva, A., 2013. Direct numerical simulation of a NACA0012 in full stall. *Int. J. of Heat and Fluid Flow* 43, 194–203.
- Shinde, G., Joshi, A., Nikam, K., 2013. Numerical investigations of the drivAer car model using opensource CFD solver OpenFOAM. In: *OSCIC 13*, Hamburg, Germany.
- Spalart, P. R., 2009. Detached-eddy simulation. *Annu. Rev. Fluid Mech.* (41), 181–202.

- Strangfeld, C., Wieser, D., Schmidt, H.-J., Wosidlo, R., Nayeri, C., Paschereit, C., 2013. Experimental study of baseline flow characteristics for the realistic car model drivAer. SAE Technical Paper (2013-01-1251).
- Trias, F. X., Lehmkuhl, O., Oliva, A., Pérez-Segarra, C. D., Verstappen, R. W. C. P., 2014. Symmetry-preserving discretization of Navier-Stokes equations on collocated unstructured grids. *J. Comp. Phys.* 258, 246–267.
- Werner, H., Wengle, H., 1991. Large-eddy simulation of turbulent flow over and around a cube in a plate channel. In: 8th Symposium on Turbulent Shear Flows, Munich, Germany. pp. 155–168.
- Wieser, D., Schmidt, H.-J., Müller, S., Strangfeld, C., Nayeri, C., Paschereit, C., 2013. Experimental comparison of the aerodynamic behavior of fast-back and notchback drivAer models. SAE Technical Paper (2013-01-1251).
- Yang, X., sadique, J., Mittal, R., Meneveau, C., 2015. Integral wall model for large eddy simulations of wall-bounded turbulent flows. *Phys. Fluids* 27 (2).