City Car Drag Reduction by Means of Shape Optimization and Add-On Devices

CITY CAR DRAG REDUCTION BY MEANS OF SHAPE OPTIMIZATION AND ADD-ON DEVICES

A. Ferraris[0000-0003-0712-3399], A.G. Airale[0000-0002-6857-1008], D. Berti Polato[0000-0001-8241-8192], A. Messana[0000-0002-2350-6149], S. Xu[0000-0001-9824-0800], P. Massai, and M. Carello[0000-0003-2322-0340]

Mechanical and Aerospace Engineering Department of Politecnico di Torino, Italy

massimiliana.carello@polito.it

Abstract

In recent years, the automotive industry has moved its atten- tion to green technologies, investing in high efficiency engines, lightweight materials, and low-rolling-resistance tires. Under these circumstances car body styling and aerodynamics play an important role in reducing vehicle drag force, permitting an improvement in fuel and energy efficiency. The objective of this study is the reduction of a city-car prototype's aerody- namic resistance by means of standard aerodynamic devices (i.e. spoiler, finlets, rear underbody, front bumper, rear dam and wheel cover). This work starts with a CFD analysis performed on the baseline configuration of the XAM 2.0 (eXtreme Automotive Mobility) vehicle, whose critical areas are then considered for the aerodynamic improvement. A CFD analysis of vehicle aerodynamics is performed in order to design differ- ent add-on features to be manufactured and tested in Pininfarina Wind tunnel. A final correlation between virtual and experimental results is carried out, validating the drag reduction, demonstrating the predictive capabilities of CFD analysis.

Keywords: Aerodynamic Features, Wind Tunnel Test, CFD Drag Reduction, City Car

1 Introduction

The aerodynamic optimization of car body shapes represents the main designing trend in the place contemporary here automotive industry. The effort to decrease the aerodynamic drag of the car is driven by a desire to reduce fuel consumption (and consequently pollutant emissions and energy requirements) [14] [3]. Furthermore, the study of the aerodynamics provides new direction to the designers of future generations of cars. For example, a reduction of 0,01 of the coefficient of drag (named CD according to [13]) corresponds to an 0.85 kilometers per liter improvement in fuel economy, as stated by [12].

The main objective of this study is the aero-drag reduction of the XAM 2.0 city-car prototype, using only add-on solutions first by means of CFD simulations and second by a final validation in real wind tunnel tests. The XAM 2.0 is a prototype entirely designed, built and developed at the Politecnico di Torino. It is designed for open road use, integrating know-how and the most successful aerodynamic features from the previous version (XAM 1.0), developed by the H2politO team to race in the urban concept category of the Shell Eco-marathon competition [5]. XAM 2.0 is a two-seat serial Extended-Range Electric Vehi- cle (E-REV) with a brushless Interior Permanent Magnet (IPM) motor and a Wankel thermal engine, homologated as heavy quadricycle SUB-A segment (L7e) [9]. The overall external size (LxWxH) is 2.880x1.300x1.280 meters with a total weight of 600 kg.

The modification of the external shape by adding devices to the model with- out changing the underlying car body is an interesting case in automotive practice, as it represents either a restyling of an existing car model or the development of a body kit purposed with the object of refining aerodynamic performances, thud brings time and cost saving benefits with respect to a complete redesigned.

2 Methodology Description

The vehicle XAM 2.0 is chosen as a case study due to its nature as a laboratory- vehicle, a test-bench for the implementation of new technologies concerning advanced materials, innovative suspension and propulsion solutions.

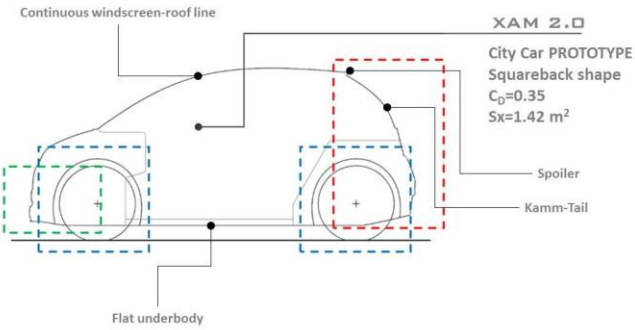

Fig. 1. Scheme of XAM 2.0 baseline model and affected-by-modifications areas (squares)

The baseline model showed some interesting aerodynamic features (Figure 1), particularly designed for drag reduction such as:

– a continuous windscreen-roof line reducing a disturbance in reduction in the uppermost area of the windshield at the intersection with the roof;

– a spoiler that helps to properly detach the airflow from the roof, limiting the wake dissipation;

– a flat underbody with covered wheel arches to avoid flow dissipative vortexes due to the mechanic system of the vehicle.

Thanks to these features the drag coefficient (CD) of the baseline model measured in the wind tunnel is equal to 0.35 (at 50 km/h speed) with a frontal area of 1.42 m2. After the preliminary analysis of the baseline model, the CAD models of the aerodynamic features (i.e. rear diffuser, front bumper, spoiler, finlets) are created using Alias (Autodesk) software, and introduced into the STAR-CCM+ virtual model, where several CFD simulations are carried out to assess the effectiveness of the introduced modifications.

Finally, the effect of the CFD-studied aerodynamic features is tested in real life: these features are produced through rapid prototyping technology and mounted on the vehicle, and a session of physical tests is performed in the Pininfarina Wind Tunnel, validating their effectiveness in drag-reduction.

3 CFD Preliminary Analysis

The CFD approach to aerodynamics engineering has rapidly grown in the last 50 years [4]. The virtual model of the vehicle XAM 2.0 has been studied using a standard CFD methodology for vehicle simulations, in order to understand which parts of the car body are the main source of the air flow deflection and vortex creation. The computational domain for CFD simulations included the virtual wind tunnel and the car model. The computational domain has been halved thanks to model symmetry, leading to a significant reduction in the simulation time [16]. In order to achieve a good convergence of the solution, several volu- metric controls zones from a minimum cell size of 8 mm to a maximum of 50 mm have been introduced. The final volume mesh model has 11 million polyhedral elements. Considering the Reynolds number at tested speeds (30, 50, 70, 120 km), the chosen physics model of the fluid domain (standard Air at reference pressure of 101325Pa, density of 1, 225 kg and dynamic viscosity of 1, 85e − 5 kg) is of segregated type with RANS-equations and Realizable K-Epsilon Two-Layer Turbulence. A good convergence of the solution with residuals under 1e − 4 was reached after 3000 iterations.

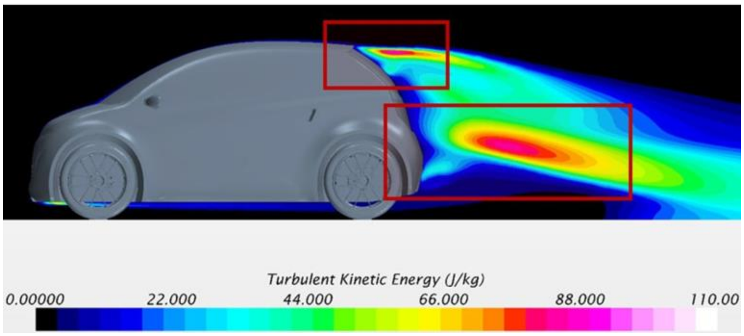

Fig. 2. Turbulent kinetic energy plot in the symmetry plane of the baseline model

The initial study underlined several disturbances on the rear-body and the front bumper. In particular, the turbulent kinetic energy plot in the symmetry plane of the virtual wind tunnel (Figure 2) showed a high deflection of the air flows leaving from the roof and the vehicle underbody, which means energy dissipation, thus increased resistance to motion. The two highly dissipative regions in correspondence with upper and central parts of the wake highlight that the angles of spoiler and diffuser can be modified to improve the wake area, with the aim of drag reduction. Moreover, the top view representation of the vorticity at the height of the car belt-line showed a high deflection of the airflow exiting the car sides This airflow behavior could be corrected by introducing properly designed side spoilers (i.e. finlets) to create a proper edge for flow detachment and avoid deflection and vortex creation.

Starting from the preliminary CFD simulations of the baseline model, the regions of the vehicle mainly involved in aerodynamic dissipation are the rear-end and the front bumper. After a benchmark analysis the most successful aerodynamic solutions applied on road vehicles, the proposed features are rear diffuser, frontal bumper, spoiler, finlets, lokari dams and wheel shields. Note that last two are tested only in the real wind tunnel. The first serves to avoid the under-body flow into the wheel case and so wheels disturbances, whereas wheel shield reduce the turbulence generated by the wheel rotation.

As the main objective of the project is aerodynamic drag reduction, particular attention is paid to the drag coefficient. A histogram displaying the reduction attributed to each feature in virtual testing is reported in Figure 3. The total reduction achieved with CFD simulations is about 7% with respect to the baseline. The positive results obtained from the CFD simulations justified the decision to manufacture real versions of the studied features.

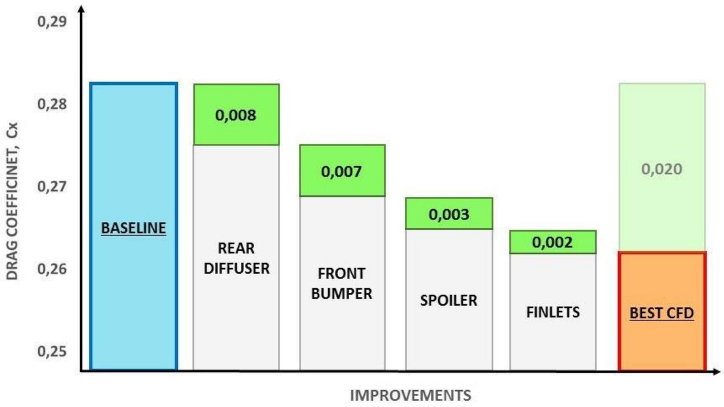

Fig. 3. CFD drag reduction for each aerodynamic studied solution

The XAM 2.0 baseline model has an inclined rear diffuser of about 11◦, with respect to the horizontal plane, at the final part of the underbody. In order to reduce the momentum of the flow (shown in Figure 2) and also the drag [11][2], a lower diffuser inclination is required. The best result is achieved with a diffuser slope of 5◦ where the geometry modification is represented. The positive effect of this solution is shown by the turbulent kinetic energy distribution in Figure 4, where the modified diffuser is compared with the baseline model.

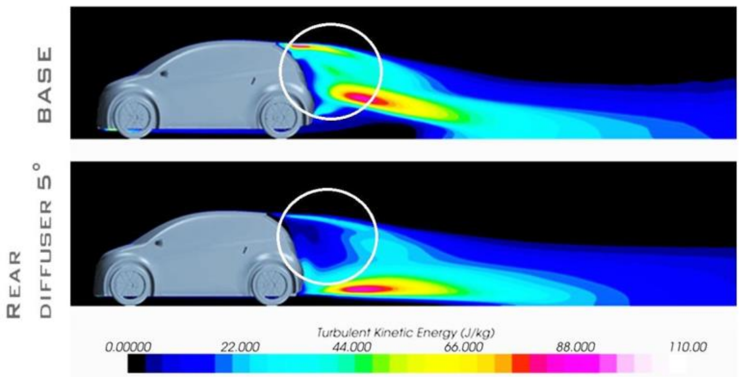

Fig. 4. Turbulent kinetic energy distribution of baseline model (top) and modified diffuser (bottom) in symmetry plane

The front bumper is a critical region for passenger car aerodynamics, since it affects the airflow pattern in the underbody and, thus, the wake. The baseline model featured an edge corresponding with the bumper [2][10].

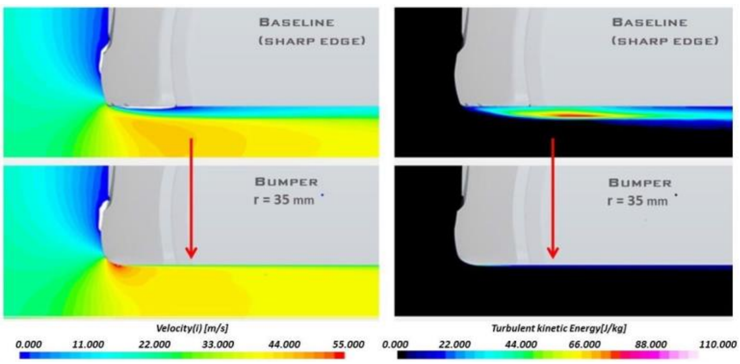

Fig. 5. Velocity (left) and turbulent kinetic energy (right) distribution comparison of the bumper cross section

The CFD analysis showed a noticeable recirculation in this region and this is highlighted by the velocity and turbulent kinetic energy distribution in the bumper cross section (Figure 5 on the top). It was concluded that the edge was responsible for the flow detachment, so a rounded bumper with a 35 mm radius was analyzed and substituted into the model in order to reduce the aero-drag.

A spoiler aimed at aero-drag reduction consists of an elongation of the roof surface over the rear window. According to previous studies available in literature, the roof angle should be about 10◦ with respect to the horizontal in order to avoid deflection [15][1].

Different configurations with a 100 mm length and three inclination angles (6◦, 9◦, 12◦ downward) have been studied.

For the CFD analysis of the effect of the proposed spoiler configurations, in terms of resistance reduction, the choice of the best spoiler configuration is driven by the drag coefficient values: the 9◦ spoiler has the greatest drag lowering effect.

In Figure 6 it can be seen, through the turbulent kinetic energy in the wake of the vehicle, that the introduction of a properly sloped spoiler with the right length (in the present case 9◦ and 100 mm) in the upper part of the wake affects also the lower region, contributing to reducing the vortex turbulence.

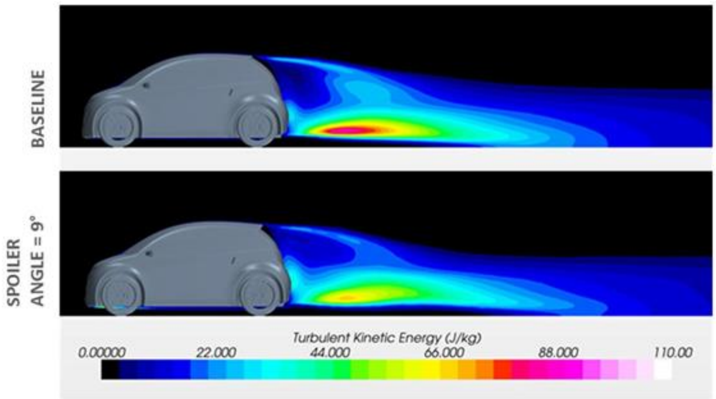

Fig. 6. Turbulent kinetic energy in the wake of the vehicle in baseline and 9◦ spoiler model

Finlets are side spoilers which create an edge to guarantee the detachment of the flow from the vehicle sides. With the aim of avoiding the airflow deflection, finlets should have a tuned length and slope with respect to the symmetry plane. Several finlets configurations with a length equal to 20 mm or 40 mm and different angles have been studied in this project: 0°, 4° and 8° inward.

By analyzing the vorticity plot in a cross section 600 mm above the wheel axis it was noticed that the 20 mm long finlet did not provide benefits to the model, then attention is focused on the 40 mm long finlets. The one angled 4° inward showed the shortest and smallest disturbed wake region.

4 WIND TUNNEL TESTS

Regarding the production of the CFD-studied features, different technologies and materials were chosen, considering the best trade off between mechanical resistance, cost, weight, quality and shape. I.e. the bumper has been made with carbon-fiber material because of its dimension, location and geometry, whereas the spoiler, finlets and lokari dams have been made in ABS material using rapid- prototyping technology, which assured the suitable properties with lower cost. The diffuser, instead, is made from a milled block of foam with the fissures then filled. After the production phase, all the features are painted red to highlight them with respect to the rest of the car (Figure 7).

Care is taken to ensure the mount could be easily manipulated, thus the devices could be added or swapped during the wind-tunnel test day. The spoiler, the finlets and the lokari dams are divided into different parts in order to be easily replaced. The spoiler was split into a fixed base and a replaceable part, whereas the finlets consisted of a fixed base, a replaceable surface and a triangular connector to the spoiler.

Fig. 7. Aerodynamic features (from left to right: bumper, spoiler and finlets, lokari dams, diffuser) mounted on the vehicle in wind tunnel test

Moreover, some different approaches are used to mount these new aerodynamic components, i.e. adhesive tape (for bumper, spoiler and diffuser) or screws (spoiler, finlets, lokari dams, wheel shields), depending on their location and weight.

Wind tunnel tests have been performed in the Pininfarina wind tunnel. Its floor includes a particular moving belt called T-belt [7, 8]. It is called this due to the shape that is formed by a central belt and two lateral belts in the front of the car. These reproduce the ground effect condition seen in real world conditions. The wheels are positioned on rollers that are connected to a three-axes dynamometer. This provides the three components of the force induced on the vehicle by aerodynamic effects (lift, drag and skin friction) and exchanged with the ground through the wheels.

The tests are carried out at three different speeds (30, 50 and 70 km/h), using a 14-holes probe at a distance of 150 mm behind the vehicle for data acquisition, As done for the CFD study, in wind tunnel each aerodynamic feature is applied sequentially in addition to the best previous, configuration. Overall 51 configurations of the vehicle with aerodynamic features are tested, always using an acquisition data time of 40 seconds, with activated BSS (Basic Suction System) and T-Belt devices. TGS (Turbulence Generation System) device, instead, is off. The wind tunnel test at 70 km/h results are reported in Figure 8. In red the best result of the vehicle equipped with rear diffuser, bumper, spoiler and finlets

is highlighted, which is the final layout tested with CFD.

Comparing CFD (Figure 3) and wind tunnel data (Figure 8), a difference in absolute results is found to be in the order of 20% - 25%. Furthermore experimental results do not confirm the solution studied with CFD, in fact correlation differences have been reported for spoiler and finlets angles.

The best solution found in the real and virtual case is:

– CFD: Spoiler at 6° + Finlets at 4° + Bumper + Rear Diffuser

– Wind Tunnel: Spoiler at 9◦ + Finlets at 8◦ + Bumper + Rear Diffuser

Despite the reported differences in best solution angle, the strong point of per- formed correlation is the relation between the CD relative improvement of each feature. In fact as shown in Table 1 the difference in percentage improvement obtained by each feature between CFD and Wind tunnel, is always lower than 1%.

Table 1. CFD - WT correlation of percentual CD reduction

Table 1. CFD - WT correlation of percentual C_D reduction

AERO FEATURE	CFD	WIND TUNNEL	ΔCFD-WT
REAR DIFFUSER	-2.85%	-3.12%	0.27
FRONT BUMPER	-2.56%	-2.64%	0.09
SPOILER	-1.13% (at 6°)	-0.90% (at 9°)	0.23
FINLETS	-0.76% (at 4°)	-1.21% (at 8°)	0.45
TOTAL	**-7.30%**	**-7.86%**	**0.56**

Fig. 8. Wind Tunnel best results at 70 km/h

5 CONCLUSIONS

The present study shows how the development of aerodynamic features con- tributes to the drag reduction. The obtained drag reduction is due to the use of standard aerodynamic features which consist of add-on features which have been applied on the vehicle once the external shape is already defined. Further improvement in aerodynamics could be achieved with the integration of stylish shapes and aerodynamic functions by means of active or passive flow control devices, considered in the original design process of the car [5].

The studied features permit a global CD reduction of more than 7% in the best correlated layout, corresponding to a fuel consumption reduction of around 2 kilometers per liter [12]. Analyzing results of the virtual and wind tunnel ex- periments, the main differences in absolute value of CD are evident only with respect to the spoiler and finlets angles. Despite these differences, an interest- ing point is the high level of correlation obtained, considering the percentage improvement given by each feature, previously designed with CFD and then validated by the wind tunnel. A further investigation of the CFD model could be performed in order to obtain a better correlation and improved flow analysis, i.e. testing different turbulence models (DES, LES as reported in [6]) and mesh refining in critical areas, increasing computational effort as drawback.

Acknowledgements

Special thanks to the hard work of all the Aerodynamics group, especially to Andrea Serra and to Beta-CAE, Star-CCM+ and Pininfarina for their support.

References

1. Altaf A, Omar AA, Asrar W (2014) Passive drag reduction of square back road vehicles. Journal of Wind Engineering and Industrial Aerodynamics 134: 30-43. doi:10.1016/j.jweia.2014.08.006

2. Barlow JB, Guterres R, Ranzenbach R (2001) Experimental parametric study of rectangular bodies with radiused edges in ground effect. Journal of Wind En- gineering and Industrial Aerodynamics 89(14-15): 1291-1309. doi:10.1016/S0167- 6105(01)00146-5

3. Barnard RH (2001) Road vehicle aerodynamic design: an introduction. Mechaero, Hertfordshire. OCLC: 802928257

4. Blocken B, Toparlar Y (2015) A following car influences cyclist drag: CFD simula- tions and wind tunnel measurements. 145:178-186. doi:10.1016/j.jweia.2015.06.015

5. Carello M, Serra A, Airale, AG, Ferraris A (2015) Design the City vehicle XAM using CFD Analysis. SAE World Congress, Detroit 21-23 April, pp. 7. doi:10.4271/2015-01-1533

6. Celik I (2005) RANS/LES/DES/DNS: The Future Prospects of Turbulence Mod- eling. Journal of Fluids Engineering 127(5): 829. doi:10.1115/1.2033011

7. Cogotti A (1995) Ground effect simulation for full-scale cars in the pininfarina wind tunnel. doi:10.4271/950996

8. Cogotti A (2007) The new moving ground system of the pininfarina wind tunnel. doi:10.4271/2007-01-1044.

9. Commission, E.U. (2014) Commission delegated regulation (EU) no 44/2014 of 21 November 2013 supplementing regulation (EU) no 168/2013 of the European parliament and of the council with regard to the vehicle construction and general requirements for the approval of two- or three-wheel vehicles and quadricycles text with EEA relevance. Official Journal of the European Union.

10. Guilmineau E (2008) Computational study of flow around a simplified car body. Journal of Wind Engineering and Industrial Aerodynamics 96(6-7): 1207-1217. doi:10.1016/j.jweia.2007.06.041

11. Guilmineau E (2014) Numerical Simulations of Flow around a Realistic Generic Car Model. SAE International Journal of Passenger Cars - Mechanical Systems 7(2): 646-653.

12. Shenkel in the book of Lance MH (2012) The shape of green: aesthetics, ecology, and design. pp 37

13. International SAE (2010) j1594 vehicle aerodynamics terminology.

14. Schuetz T (2016) Aerodynamics of road vehicles.

15. Thacker A, Aubrun S, Leroy A, Devinant P (2012) Effects of suppressing the 3D separation on the rear slant on the flow structures around an Ahmed body. Journal of Wind Engineering and Industrial Aerodynamics. 107-108: 237-243. doi:10.1016/j.jweia.2012.04.022

16. Skaperdas V, Iordanidis A (2013) The influence of mesh characteristics on CFD simulations for automotive applications. Automotive Simulation World Congress, October 29-30, Frankfurt, Germany.