Available online at www.sciencedirect.com

ScienceDirect

Procedia Manufacturing 30 (2019) 685–692

Procedia MANUFACTURING

www.elsevier.com/locate/procedia

14th Global Congress on Manufacturing and Management (GCMM-2018)

Windscreen angle and Hood inclination optimization for drag reduction in cars

Vignesh S[a*], Vikas Shridhar Gangad[a], Jishnu V[a], Maheswarreddy[a], Amal Krishna[a], Yagna S Mukkamala[b**]

[a]*M.Tech Automotive Engineering, VIT University, Vellore 632014, India*
[b]*Professor, SMEC, VIT University, Vellore 632014, India*

Abstract

Drag reduction in cars is evident by the angle of windscreen inclination. Drag reduction helps to reduce emissions. Windscreen inclination affects the height of the vehicle. The curvature of the windscreen should be optimized in order to avoid the cross-wind effect. The dimensions of the windscreen of a car are obtained from works of literature and modelled in modelling software. Firstly, the effect on windscreen inclination with respect to drag is studied using ANSYS Fluent. CFD analysis is done with inlet velocity of 25 m/s and the influence of vehicle height against the drag coefficient is identified. By varying inlet velocity of air, the pressure contours in the frontal hood of the vehicle are studied. Hood inclination should be optimized for minimum drag. From the studies, it is observed that 32 ° back windscreen angle gives the minimum C_D. The front windscreen angle should be below 45° for minimum drag and C_D is decreasing linearly as hood inclination increases. The angle of windscreen inclination proposed such that it does not affect driver visibility. It should accumulate the raindrops and dirt effectively. Windscreen inclination angle selected such that it does not affect the stability due to change in vehicle dimensions. The drag coefficient is reduced by optimizing the frontal area. The windscreen inclination is obtained by taking visibility concerns into account. From the studies, It is observed that front windscreen angle 60°, back windscreen angle 32° and 15° hood inclination gives the minimum C_D as 0.320.

© 2019 The Authors. Published by Elsevier Ltd.
This is an open access article under the CC BY-NC-ND license (https://creativecommons.org/licenses/by-nc-nd/4.0/)
Selection and peer-review under responsibility of the scientific committee of the 14th Global Congress on Manufacturing and Management (GCMM-2018).

Keywords: Coefficient of drag; Windscreen angle; Hood inclination; ANSYS fluent

*Vignesh.S . Mob. +91 8300151176
E-mail address: vignesh.s2017@vitstudent.ac.in

**Yagna S Mukkamala. Tel: 91-0416-2202203
E-mail address: yagnasmukkamala@vit.ac.in

2351-9789 © 2019 The Authors. Published by Elsevier Ltd.
This is an open access article under the CC BY-NC-ND license (https://creativecommons.org/licenses/by-nc-nd/4.0/)
Selection and peer-review under responsibility of the scientific committee of the 14th Global Congress on Manufacturing and Management (GCMM-2018).
10.1016/j.promfg.2019.02.062

1. Introduction

Research is going on in the field of the automobile for the past many years to improve the efficiency. Fuel economy is one of the major concerns faced by internal combustion engines. Due to the high demands for the fossil fuels, designers were primarily focused on improving the efficiency through minimizing engine losses. However, improving aerodynamics of a car remained secondary aspect till then [1]. Further developments in technologies to improve the efficiency of the engine have almost reached its saturation state. The research now focuses on improving the aerodynamic aspects of the vehicle to enhance its performance.

Design challenges of an automobile are the compromise between aerodynamics and the ergonomics. Required shape for an automobile is restricted due to the ergonomics [2]. The resistance forces experienced by a moving vehicle on a level road are rolling resistance and aerodynamic drag; both seem to have some influence at higher speeds. Aerodynamic drag coefficient is the important factor responsible for the drag experienced by the vehicle. Computational fluid dynamics (CFD) is developed by pure mechanics, numerical computation methods and computer technologies [3]. The applications of computational fluid dynamics in vehicle body design were in use since the 1960s. This method makes use of solving problems related to the aerodynamics of automobiles. Air resistance reduction not only helps in the speed improvement of the vehicle but it also reduces the fuel consumption and improves the stability of the vehicle [4]. As emission norms are becoming stringent, reduction in aerodynamic drag results in decreased emissions, in turn, improves the efficiency. In the designer's point of view, only the drag coefficient (C_D) and frontal area (A) can be controlled. Designing the vehicle considering the frontal area will result in the reduction of the aerodynamic drag force [5].

Nomenclature

A	Frontal area
CFD	Computational Fluid Dynamics
C_D	Coefficient of drag
CO_2	Carbon dioxide
F_d	Drag force
PVB	Poly Vinyl Butryl

2. Literature survey

R.H. Heald concluded that the reduction in fuel consumption due to the better aerodynamic design of car reduces the emissions from the vehicle [1]. J. XU et.al carried out strength related investigations in the windshield and its aerodynamic perspective windscreen is the important part of the vehicle in pedestrian safety. It suggests that different material of windscreen, which gives the least injury to pedestrian and provides enough impact strength [2]. Mateusz Wasik et al commented on windscreen inclination has some limitation due to its effect on other factors such as visibility, dirt and water accumulation ability [3]. Darko et.al investigated on the windscreen inclination and angle formed by hood about their respective contributions in aerodynamic resistance. The car is modeled in Autodesk 3DS Max and analyzed in ANSYS Fluent. The angle between the frontal hood and windscreen is changed for the betterment of aerodynamic properties. The main theme of this work is the reduction of air resistance [4]. Jilesen, J et al. shared about Water flow over the windscreen during raining is studied with Jaguar saloon model. Numerical simulations are carried out and a commercially available wind tunnel Lattice Boltzmann solver is used in CFD for solving thermal model [5]. Zhu Hui et al. discussed windscreen inclination being the important parameter in drag reduction and which gives direct effect on the fuel consumption by the vehicle [6]. He Xuhui et.al proved that deflector used to eliminate high drag regime to low drag regime which can lead to a decrease in aerodynamic drag. The curvature of the windscreen should be optimum to avoid cross wind effect [7]. A. Varun et al discussed about the laminated windshield of the car analysis using the finite element method. Importance is given to windscreen material PVB which is sandwiched between two layers of glass. Purpose of this work is to analyze the impact strength of windscreen [8]. Ferziger J.H et al investigated about Basics behind the computational flow analysis over external surface. Windscreen flow analysis is external one and thus the physics behind momentum equations in noticed [9]. D. Ramasamy et al. have performed aerodynamic analysis of proton hybrid car [10]. Hence, the aerodynamic evaluation of the same car by changing windscreen and hood angle for further drag reduction is carried out in this paper.

3. Geometry modeling

CATIA V5 R17 was used for modeling the car. Proton Iswara car model is designed and drag analysis for the same is carried out. By changing the windscreen and hood angle inclination in the solid model, the coefficient of drag is estimated [10]. The specifications of the car model provided below.

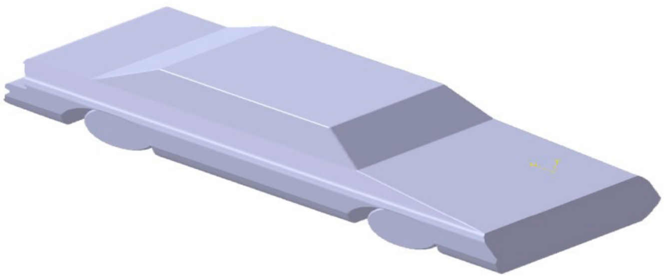

Fig. 1 Proton Iswara car

Table 1. Specifications of Proton Iswara car

Specification	Data
Maximum Height	1360 mm
Maximum width	1655 mm
Maximum length	4110 mm
Wheelbase	2380 mm
Track (front and rear)	1406 mm & 1356 mm
Gross vehicle weight	1360 Kg
Maximum Output	62 KN @ 6000 rpm
Maximum Torque	109 Nm @ 4000 rpm

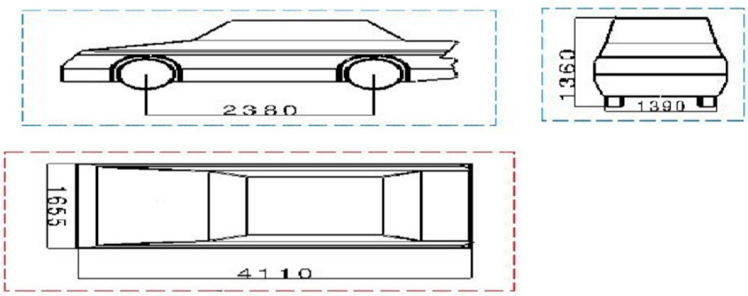

Fig. 2 Orthogonal views

4. Analysis in ANSYS Fluent

The modeled geometry is imported in ANSYS workbench. Design Modeler is used to create named selections such as inlet, outlet, wall etc.

4.1. Creating an Enclosure & Boolean

For simulating the airflow around a car, a fluid volume box is created. This box encompasses the car as the air domain and subtraction of car body from the fluid volume box is done by the Boolean function. Domain size is taken as 5:1 ratio of the car dimensions and placed in the symmetric plane. The computation domain was a rectangular box of 4137.5 mm x 10275 mm x 3400 mm based on symmetry conditions.

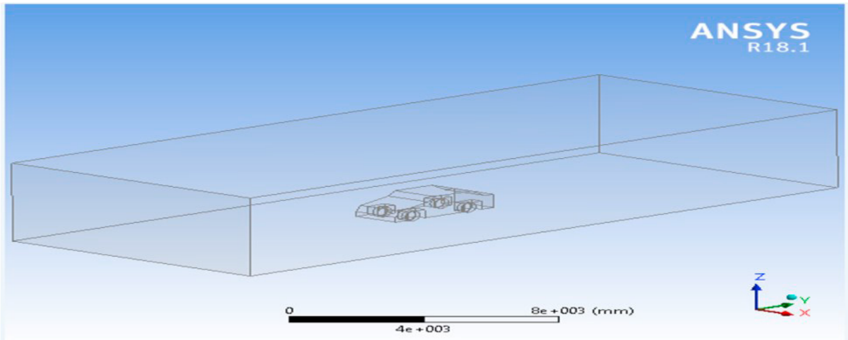

Fig. 3 Domain

4.2. Surface meshing

During the mesh generation, certain surfaces need a sizing function to obtain a refined mesh in order to achieve an accurate drag. Hence tetrahedral mesh is used. Mesh shaping is given through curvature and fine refinement is selected for accuracy in meshing.

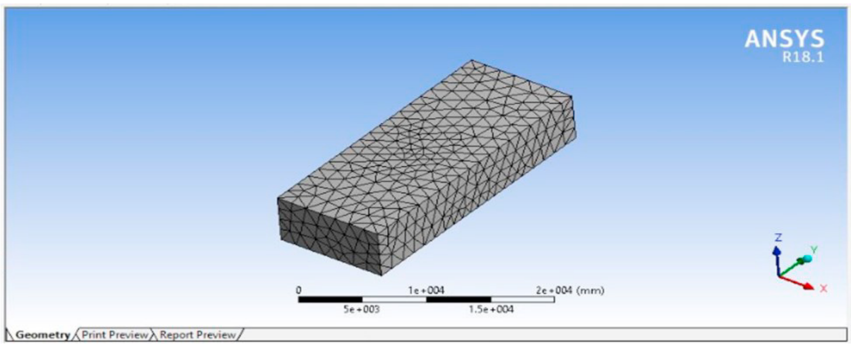

Fig. 4 Meshing

4.3. Model setup

The analysis type incorporated here is pressure based steady-state solver. It is a viscous model & Laminar flow is preferred. In Boundary conditions, the car geometry is kept on the stationary wall with no friction condition. Input velocity is 25 m/s. From the Reference values, the velocity of flow should be changed corresponding to input velocity.

4.4. Solver

The governing equations involved in a typical computational analysis include Continuity equation, Momentum equation and Energy equation. In this analysis heat transfer mechanics is not considered. Therefore while solving aerodynamic problems, Continuity equation and momentum equation is sufficient. Coming to solution step, the solver preferred for this analysis is Coupled solver with both turbulent kinetic energy and intensity as second-order up winding scheme. In the Monitors section, the flow currant number is varied as the solution progresses.

4.5. Initialization

Hybrid initialization is done for e-6 convergence as the initial condition for solving at time t=0. The number of iterations is varied till the convergence has reached. By varying the hood inclination, frontal and back windscreen angle; the coefficient of drag is estimated. The windscreen is optimized based on minimum drag obtained from various combinations of hood angle, front and back windscreen angle.

5. Results and Discussion

5.1. C_D of changing back windshield angle

The back windshield angle has direct influence on flow separation at the rear of a car, which in turn possesses greater influence on the drag coefficient C_D. The size of passenger compartment is considered in accordance with back windshield angle. Results are simulated at every 5° from back windshield angle from 10°-90°. In order to get maximum and minimum C_D more precisely, more simulation point were added around back windshield from 35°to 45°. A maximum C_D was achieved at back windshield angle 35°. The total result of C_D shows in figure 5.

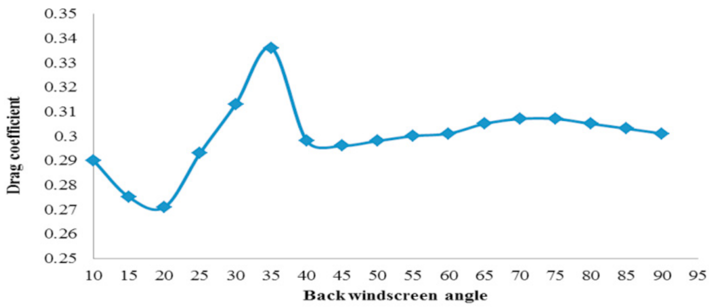

Fig. 5 Effect of back windscreen angle on the drag coefficient

The aerodynamic drag is dependent on pressure differential resistance (90%) and friction resistance (10%). While analyzing the aerodynamic drag created by both pressure differential resistance and friction resistance due to the change of back windshield angle the problem becomes more complex.

For back windshield angle 10° - 15° the positive pressure area at the rear of the car increases which results in the decrease of C_D. At back windshield angle 20° - 35°, Positive pressure area on the rear of the car increases while the positive area on the luggage remains unchanged which result in the increase in C_D. If the back windshield angle lies in between 35° - 45°, Negative pressure area at the rear of the car decreases and positive pressure area at rear of the car decreases which results in the decrease of C_D and for back windshield angle which lies in between 45° - 95°, Tail pressure magnitude basically does not change and positive pressure area increases which results in almost no change in C_D.

5.2. C_D of changing front windshield angle

The air has a high velocity on the hood before it reaches the front windshield. Separation region is created at the bottom of the front windshield when the flow reaches the front shield and slowdown. The C_D of numerical simulation of changing front wind shield angle shown in figure 6.

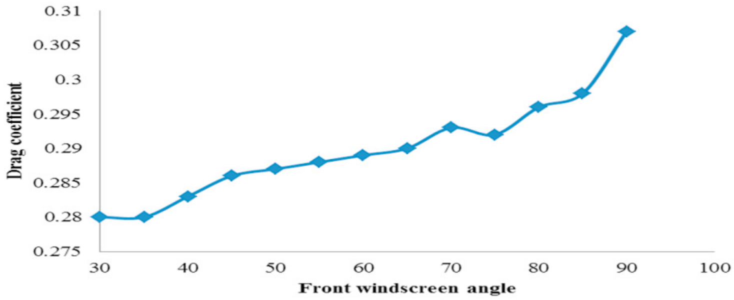

Fig. 6 Effect of front windscreen angle on drag coefficient

Small changes in C_D were observed when the front windshield angle is below 45° if it is greater than 45° C_D increases with the increase of front windshield angle.

When the front windshield angle is less than 45°, the separation area is bigger at the bottom of the front windshield, as a result, the area in between the front windshield and roof increases with constant C_D. When the front windshield angle lies in between 45°- 55°, the separation area is bigger as same as that of the previous case and as a result, the area increases with increase in C_D. When the front windshield angle is greater than 55° the lines of separation and reattachment were far away from each other.

5.3. C_D of changing hood angle

The C_D of numerical simulation of changing the hood angle is shown in the figure. C_D decreases with increase in hood angle due to the area is very less at the frontal part below the hood. Hence C_D decreases linearly.

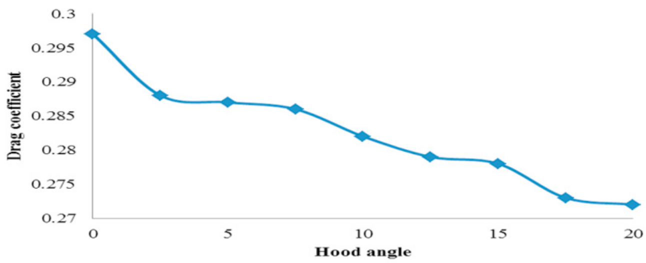

Fig. 7 Effect of hood angle on drag coefficient

5.4. Orthogonal experiment analysis

According to the cases, which are discussed above, an orthogonal experimental analysis is carried out by considering 6 cases and the corresponding C_D is tabulated.

Table 2. Orthogonal experimental analysis

Case	A(front windshield)	B (back windshield)	C(hood)	C_D
1	30	35	10	0.400
2	30	38	15	0.390
3	45	32	10	0.370
4	45	35	15	0.335
5	60	32	15	0.320
6	60	38	15	0.335

5.5. Pressure contour

From the pressure contour distribution, it is observed that stagnation at the front end of the car results in increased pressure. Also with the increase in front windscreen angle pressure drag is increasing, 60° windscreen inclination causes more pressure drag.

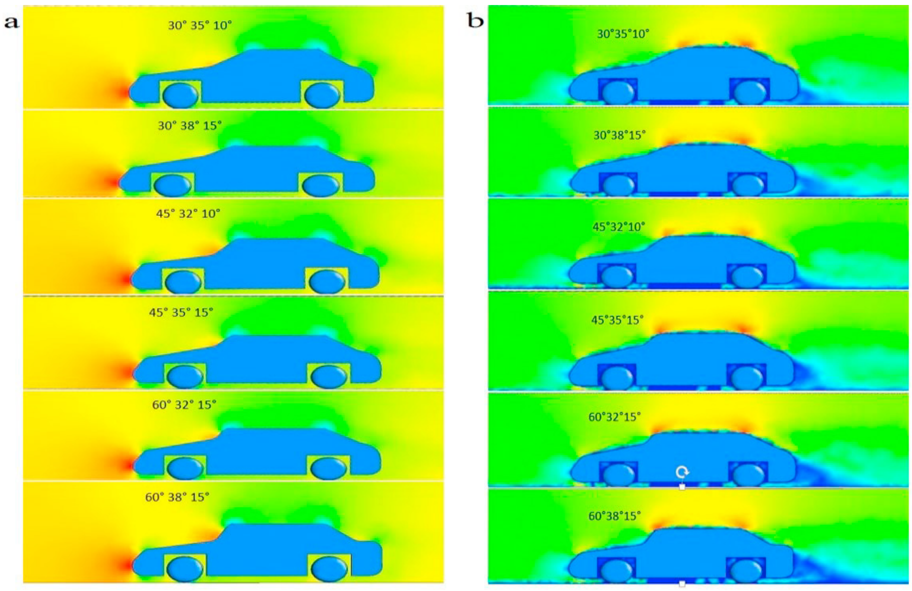

Fig. 8 (a) Pressure contour; (b) Velocity contour

5.6. Velocity contour

Viscous force is predominant considering the velocity distribution, air while passing through car get attached to the body thereby creating more viscous drag. Viscous drag is less significant regarding the total drag faced by the car. Pressure drag serves as the predominant drag in designing a vehicle aerodynamically.

6. Conclusion

Maximum C_D at back windshield angle is observed at 35° in Proton Iswara car. Minimum C_D at back windshield angle is obtained at 15°. Small changes in C_D were observed when the front windshield angle is less than 45°. If it is greater than 45° C_D increases. C_D decreases along with the increase of hood angle. The orthogonal experiment is effective for estimating aerodynamic drag and it saves time and computing resources. From the results, at the front windscreen angle 60°, rear windscreen angle 32° sand hood inclination angle of 15°. This combination of angles for Proton Iswara car possesses a drag coefficient of 0.320. For a sedan, the drag coefficient is 0.44 [12]. The aerodynamically modified design reduced drag coefficient by 27 %. This is helpful in reducing the drag resistance, which in turn consumes less fuel, thereby less emission as well.

Acknowledgements

The authors would like to thank Prof. Yagna S Mukkamala, SMEC, VIT University for his valuable guidance throughout the project duration.

References

[1] R.H. Heald, Aerodynamic Characteristics of Automobile Models, U.S. Department of Commerce, Brueau of Standards Research Paper RP591, Part of Brueau of Standards Journal of Research , Vol.11, August 1933
[2] J. XU, Y.B. Li, X. Chen, D.Y. GE, B.H. Liu, M.Y. Zhu, T.H. Park, Automotive Windshield – Pedestrian Head Impact: Energy Absorption capability of inter layer material, International Journal of Automotive Technology, Vol.12, No.5 pp. 687-695 (2011).
[3] Mateusz Wasik, WOjciechSkarka, Influence of the windscreens inclination angle on the aerodynamic drag coefficient of the cars designed for the race shell eco-marathon based on Numerical simulations, Proceedings of the Institute of Vehicles 3(103)/2015
[4] Darko Damjanović, Dražan Kozak, Marija Živić, Željko Ivandić, Tomislav Baškarić, CFD analysis of the concept car in order to improve aerodynamics, International Scientific and Expert Conference TEAM 2010.
[5] Jilesen J, Spruss I, Kuthada T, Wiedemann J, Advances in Modeling A-Pillar Water Overflow, SAE Technical Paper 2015-01-1549, 2015, doi:10.4271/2015-01-1549
[6] Zhu hui, Yu huo, Numerical Analysis on Effect of Back/Front windshield and hood angle on automotive aerodynamic drag, IEEE International conference (ICISCE 2012)
[7] He Xuhui, Wang Hanfeng, Zhou Yu, Zou Chao, Aerodynamic drag reduction of an ahmed body based on deflectors, J.Wind Eng. Ind. Aerodyn. 148(2016) 34-44.
[8] A.Varun, CG Nanda kumar, Strength Investigations on Laminated windshield, IJIRSET – Vol.4 Special issue 12, Sep.2015
[9] Ferziger J.H., Peric M, Computational Methods for Fluid Dynamics, Springer, New York, 2002.
[10] D.Ramasamy, K. Mohanesan, K.Kadirgama, M.M. Noor, M.M. Rahman, Hybrid electric vehicle car body drag analysis using computational fluid dynamics, International Journal of Automotive and Mechanical Engineering, Volume 4, Issue 3, 4496-4507, September 2017.
[11] L.Parras, F.J. Bello-Millà, T.Mäkelä, C. del pino, C. Ferrera Experimental study on Ahmed body drag coefficient for different yaw angle, J.Wind Eng. Ind. Aerodyn. 157 (2016) 140-144
[12] Vivek yakkundi, S.S. Mantha, Vivek sunnapwar, Hatch back versus sedan: A review of drag issues, SSRG-IJME, volume4, issue 1, January 2017.
[13] John D. Anderson, Computational Fluid Dynamics: The basics with applications, McGraw-Hill, Inc., New York, 1995.
[14] Zhu hui, Wang Jial, Numerical analysis of effect of vehicle length on Automotive aerodynamic drag, IEEE International conference (ICISCE 2012)
[15] Jamal Naser, FirozAlam, Computational fluid dynamics study of vehicle A-pillar aero-acoustics, Applied Acoustics 74 (2013) 882–896
[16] Popat BC. Study of flow and noise generation from car A-pillars. PhD thesis, Imperial College, London; 1991
[17] WH Hucho, Aerodynamics of Road vehicles, SAE 1998
[18] Tao Jiang, Design and Simulation of Car windshield Broken Mechanism based on ABAQUS, 4th International Conference on Mechanical Materials and Manufacturing Engineering (MMME 2016).
[19] Ribaldone, Enrico, Cogotti, Francesca, Tregnago, Roberto, Riu, Erica Ruo, Scantamburlo, Giuseppe, Application ofmulti-objective optimisation to the aerodynamic developmentof passenger cars at FIAT, 8th MIRA International Vehicle Aerodynamics Conference 2010
[20] M. Timmel, S. Kolling, P. Osterrieder, and P.A. Du Bois, A Finite Element Model for Impact Simulation with Laminated Glass, International Journal of Impact Engineering, 34, pp 1465-1478, 2007.
[21] Wang.Li, Study on the Aerodynamic Characteristics of Fastback Passenger car, Chang chun: Jilin University, 2005.
[22] WANG Jia, YANG Zhigang, ZHU Hui, Grid Method Research for Numerical Simulation of Aerodynamic Drag on Notchback MIRA Model, Journal of Jiamusi University (Natural Science Edition), Vol. 30(3), 2012, pp. 652-656.
[23] Jeffrey Hoffman, Bill Martindale, Stephen Arnette, Effect of Test Section Configuration on Aerodynamic Drag Measurements, SAE paper 2001-01-0631
[24] A. Luca, M. Vincenzo, P.V. Pier, et al, A CFD-FEM approach to study wind Aerodynamics under deformation, SAE World Congress, Vehicle Aerodynamics, pp.444-451, 2004.
[25] Z.Q. Gu, L.H. Jiang, J. Wu, and G. Fang, A computer simulation on 3-D outer flow field around a passenger car, Automotive Engineering, vol. 22, pp. 296-298, 2000
[26] Bayraktar, I, Landman, D, Baysal, O, Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics, SAE Technical Paper 2001-01-2742.
[27] Lienhart H, Stoots C, Becker S, Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Modell). Springer, Berlin Heidelberg, pp. 323–330, 2002.