CFD analysis on effect of front windshield angle on aerodynamic drag

Essaghouri Abdellah¹, Wang Bo*
¹Master Student, State Key Laboratory of Automotive Safety and Energy, Tsinghua University, Beijing, China
*Associate Professor, State Key Laboratory of Automotive Safety and Energy, Tsinghua University, Beijing, China
¹abdl15@mails.tsinghua.edu.cn  *matedor@tsinghua.edu.cn

Abstract. The external aerodynamics plays an important role in the design process of any automotive. The whole performance of the vehicle can be improved with the help of external aerodynamics. The aerodynamic analysis nowadays is implemented in the recent research in the automotive industry to achieve better cars in terms of design and efficiency. The major objective of the present work is to find out the effect of changing the angle between the engine hood and the front windshield on reducing the car air resistance. A full scale three dimensional (BMW 3 series) sedan car model was carried out using the ALIAS AUTOSTUDIO 2016 a NURBS modeling tool with high quality surfaces, only the external shape of the car was modeled while the interior was not modeled. The ANSYS 17.0 WORKBENCH software package was used to analyse the airflow around the external shape of the car – the solutions of Reynolds Average Navier Stokes (RANS) equations has been carried out using realizable k-epsilon turbulence model (which is perfectly suitable for the automated calculation process) for the given car domain. In this work, the boundary layer, mesh quality, and turbulent value simulation has been compared and discussed in the result section. Finally the optimal model was selected and the redesigned car was analysed to verify the results.

1. Introduction
Nowadays with the growth of automotive industry, the aerodynamic analysis of vehicles has helped in reducing considerably the drag force acting on the external shape of the vehicles. The reduction of the air resistance not only improves the performance of vehicles in terms of the speed as it moves forward but also improves its stability and fuel consumption that is responsible for increasing CO2 emission which have a drastic effects on environment. It is only the drag coefficient (Cd) times the frontal area (A) that designer can control. [1] Thus, designing the vehicles aerodynamically helps in reducing the coefficient of drag, thereby reducing the drag force (Fd). In terms of vehicle efficiency the wind tunnel test has been playing an important role in the recent research and development in automotive manufacturers. Recent growth of the computational power machines, the computational fluid dynamics (CFD) helps to study the behavior of the flow around the vehicle with the same efficiency as the wind tunnel test without the need to create a physical model which gives a sense of relaxation to automotive design engineers, and the overall costs are greatly reduced while new and creative ideas can be easily developed and tested on virtual platform with almost the same accuracy as the wind tunnel will provide.
1.1. Objectives
In this work, our CFD analysis will be focusing on a BMW 3 series sedan car, which has been selected due to satisfaction of the following criteria:
1. Vehicle availability in almost all the world
2. The availability of the car blueprints to create the three dimensional model
3. The value of the car drag coefficient (Cd) is provided officially which will help to check the accuracy of the CFD test results. (The drag coefficient of the BMW 3 series is 0.30). [10]

1.2. Description of the study case
The aim of this simulation is to study the air flow around the car and obtain an approximate value of its aerodynamic drag. In order to reduce the global time needed in the 3D simulation and present different optimization methods, a 2D simulation was performed to study the effect of increasing the angle between engine hood and front windshield on the aerodynamic drag. The 2D simulation can be very helpful guideline in the aerodynamic design process. Two different optimization methods were investigated in this project, then the results were compared and the existing three dimensional model was redesigned with better aerodynamic characteristics.

In the first case method only the angle between the hood and front windshield was increased, while in the second case method not only the angle was increased but also the height of the vehicle was modified. It is expected that modifying the shape of the car in terms of increasing the angle between the front windshield and engine hood with respect to decreasing the height of the roof and rear side of the car without altering the rear windshield angle as shown in fig.1 (2nd case) will lead to a significant reduction of Cd as compared to the first method. In each case the angle changes by one degree (roughly), with a total of 8 models for each case have been simulated and compared in the result section. The total reduction on the car height in the 2nd case was about 6.7 cm.

![Figure 1. 2D model of the first case (left) and the second Case (right).](image)

2. Methodology
The computational fluid dynamics (CFD) analysis consist of three major tasks after the identifying the problem and the goals which are Pre-Processing, solving and Post processing.
Pre-processing: Preprocessing for CFD analysis is the step where defining all the task that take place before the numerical solutions, this includes geometry (creating the 3D model), surface mesh (mesh size and meshing process) and setup of boundary conditions. In this work, ANSYS Workbench is used in the preprocessing step.

Processing: or solution process involves solving numerical equations of fluid flow, and defining the analysis type, which is a pressure based steady state solver. In this step Fluent 17 is our solver.
Post processing: After getting the solution from the solver, the result can be viewed and analyzed in both forms numerically and graphically.

3. Processing
3.1. Geometry modeling
The car geometry should be highly detailed within all areas as compared to the final production shape. Alias Auto-studio 2016 was used for modeling the three dimensional geometry. The modeling process
includes importing the blueprints of vehicle into mentioned software and the surfaces were built using NURBS (Non uniform rational B-spline) modeling methods. Which are the industry leading tool for representing curves and surfaces in CAD as well as CAM.

\begin{figure}[h]
    \centering
    \includegraphics[width=\textwidth]{alias_autostudio}
    \caption{BMW 3D geometry in Alias Autostudio.}
\end{figure}

3.2. Creating an Enclosure
In order to simulate the air flow around the car, a fluid volume box which will encompass the car as the air domain need to be established by creating an enclosure around the car and subtracting the car body using the Boolean function. Due to the symmetric design of the car, only half of the domain was considered. The dimension of the enclosure was taken as three car length in the front, five car length between the rear and end of the enclosure and two car length in the sides. The car Length is 4.580 m.

3.3. Surface Meshing
In the mesh generation, certain surfaces need sizing function to obtain a refine mesh in order to achieve an accurate drag and lift parameters. Three bodies of local refinements were added to assure that the majority of elements will be located close to the car and in the wake region behind the car (see fig.3). To achieve a good alignment with the flow near wall boundaries since it has a significant effect on coefficient of drag, 5 layers of inflation were added to the car surface with a first aspect ratio and 1.2 growth rate as shown in fig.5.

\begin{figure}[h]
    \centering
    \includegraphics[width=\textwidth]{bodies_refinements}
    \caption{Bodies of refinements location.}
\end{figure}

\begin{figure}[h]
    \centering
    \includegraphics[width=\textwidth]{body_refinement}
    \caption{Body of refinement impact on 2D.}
\end{figure}

\begin{figure}[h]
    \centering
    \includegraphics[width=\textwidth]{tets_mesh}
    \caption{Tetrahedral mesh with and without bodies of refinement}
\end{figure}

3.4. Boundary Conditions
The boundary conditions of CFD simulation should be almost the same as the wind tunnel conditions. For this reason the enclosure inlet was named “Velocity inlet” to model the incoming flow (air) the outlet was named “Pressure outlet”. The car body and Road are stationary walls with “no slip” condition (V = 0 m/s). The symmetry plane, top and side of the enclosure all walls.
4. Theory

4.1. $k$-$\varepsilon$ model
The $k$-epsilon model is one of the most widely used turbulence models, it is by definition a two equations of the flow that includes two extra transport equations to represent the turbulent properties of the flow. Although it does not perform well due to the deficiencies of this model especially in cases of large adverse pressure gradients. [2]

4.2. Realizable $k$-$\varepsilon$ modeling
Realizable $k$-$\varepsilon$ model which is one of the most recently developed model. The realizable means that the standard $k$- is not realizable for high Reynolds number. A new $k$-$\varepsilon$ eddy viscosity model include a new formulation for the model coefficient $C_{\mu}$ which was proposed by Reynolds and Shih at al. [3] More accurate predict in terms of integral values (e.g., $C_d$) can be achieved within 2-5%.

4.3. Non-Equilibrium Wall Function
The external flow around ground vehicles has a High-Reynolds number, thus it is not practical to resolve the near wall region down to the wall which is a no-slip boundary. [4] Fluent provide useful tool to overcome this issue so called “Non-equilibrium wall functions”. The NWF not only taking into account the effect of variation in the thickness of the laminar sublayer, but also are sensitive to the effect of pressure gradients. It is recommended to use the NWFs to get more accurate prediction of vehicle aerodynamics instead of the traditional wall function for the external aerodynamic simulations.

5. Solver
In this part, the simulation was carried out using Fluent 17. The type of solver, numerical equations and the methods were used in this analysis are listed below:
- The analysis type is a **pressure based steady state solver**
- The turbulence model is a “Realizable $K$-$\varepsilon$ model” with NWFs (non-equilibrium wall functions)
- The velocity of the 2D domain $V$= 40 m/s. while the velocity of the 3D domain is $V$= 30 m/s.
The table 1. shows that the changing of the aerodynamic drag is not a linear function with the increase of the front windshield angle, in the first case the aerodynamic drag increase and reach a maximum value at the angle of 158°, this increase of the Cd is due to the increase of the separation area at the bottom of the front windshield as it can be seen from the static pressure contour (fig.6), from the angle between 160-162° the change of drag coefficient is very small and can be considered as constant due to the increase of the negative area between the front windshield and the roof. In the second case it is obvious that the drag coefficient decrease along with decreasing the height of the car, also we can notice that the pressure at the front of the car in this case is smaller than it on the first case, also the pressure underbody of the car is lower than it on the first case. From velocity contour (fig.7) it can be seen that the air flows at the front and top of the car in the 2nd case flows faster than it on the 1st case.

Table 1. The front windshield angle influence on Cd.

| Angle | Case 1          | Case 2          |
|-------|-----------------|-----------------|
| 154°  | 2.85316e-01     |                 |
| 155°  | 0.288804         | 0.284614        |
| 156°  | 0.287756         | 0.284041        |
| 157°  | 0.291805         | 0.283624        |
| 158°  | 0.292237         | 0.278222        |
| 159°  | 0.290617         | 0.276176        |
| 160°  | 0.289622         | 0.277374        |
| 161°  | 0.289269         | 0.274674        |
| 162°  | 0.287519         | 0.272041        |

Chart -1: Changing of the drag coefficient (both cases)
Three-dimensional CFD analysis

By combining the results obtained from the CFD analysis of the airflow over the 2D car and the existing 3D car model, a new 3D car model has been redesigned.

Table 2: the result of the 3D simulation

|                        | Original car | Redesigned car |
|------------------------|--------------|----------------|
| Windshield angle       | 154°         | 162°           |
| Frontal area (m²)      | 1.9602       | 1.9120         |
| Drag coefficient       | 0.28714      | 0.26173        |
| Velocity (m/s)         | 30           | 30             |

From the CFD analysis a drag of 0.28714 was obtained which indicate an acceptable agreement between wind tunnel and CFD test data. The numerical value of the aerodynamic drag for the redesigned car model which has been affected by the optimization of the external shape of the car was reduced by 8.85% as it shown in the table 2.

The solution was obtained using two stages with a coupled scheme for pressure and velocity. The first stage was performed by solving the first order upwind equations for the momentum, turbulent kinetic energy and dissipation rate, then we moved to solve the second order upwind equations. In this stage the iteration were carried up to the point where the drag coefficient value been considered negligible.

Figure 8: shows the contour cell Y plus for the realizable k-epsilon turbulence model where results are very sensitive to the value of the wall Y-plus, and it depend heavily on the first layer, to maintain the values of Y-plus between 25-300 mesh sizing functions and inflations were implemented to achieve the desired Y-plus.

Figure 9. shows the distribution of the pressure on the surface of the car, for both the existing and redesigned model it can be seen that the high pressure indicated by the red color located at the front of the car and between the hood and the front windshield, moreover it is obvious that the distribution of the pressure over the redesigned car has been reduced as expected.
From the contour of turbulent kinetic energy on the symmetric plane it can be seen that the maximum value of energy kinetic located at the rear of the car due to the flow separation, and it is obvious that the kinetic energy in the redesigned car has an upward deflection as compared to the original car.

Figure 11: shows the contour of velocity distribution on the symmetric plane and the car body surface, the blue color means a zero velocity because it was chosen as a stationary wall with no slip option, the flow over the top of the car move faster and acting downward while underbody of the car is acting upward which will produce vortex at the rear back of the car as shown in the fig.12. These two vortices are generated due to the separation of the flow which caused by the difference air pressure between the top and underbody car, and are the main cause of the reduction of the rear pressure, hence increasing the drag coefficient.
6. Conclusion
The proposed methods for the optimization of aerodynamic drag of the car by increasing the front windshield angle and shape optimization shows that the second case provide less drag also a better aerodynamic characteristic, and was confirmed by the 2D and 3D analysis of BMW 3 series sedan car which has been successfully carried out on both existing and redesigned model. The result shows great agreement with the wind tunnel test result for the original model, while the redesigned 3D model has better aerodynamics as compared to the existing model, the drag coefficient was reduced by 8.85%. The height of the car was reduced by 6.7 cm in the redesigned car. From the CFD results, it is clear that only increasing the angle between the front windshield and engine hood without altering the external shape of the car has not a big influence on the value of the aerodynamic drag. While increasing the front windshield angle and altering the shape of the car has a significant influence on the aerodynamic drag and more streamlined air flow.

Acknowledgments
The present work was supported by the State Key Laboratory of Automotive Safety and Energy, Department of Automotive Engineering, Tsinghua University. I would like to express my sincere gratitude to my supervisor Prof. Wang Bo for the continuous support of my Master’s studies and related research, for his patience, motivation, and immense knowledge. His guidance helped me in all the time of research and writing of this paper. I could not have imagined a better supervisor and mentor for my graduate studies.

References
[1] William H. Bettes, The aerodynamic drag of road vehicles: Past, Present and Future. ENGINEERING & SCIENCE / January 1982.
[2] M. Young, The Technical Writer’s Handbook. Mill Valley, CA: University Science, 1989.
[3] Shih, T.-H., Zhu, J., and Lumley, J. L. A New Reynolds stress algebraic equation model. NASA TM, 1994. K. Elissa, "Title of paper if known,” unpublished.
[4] T-H. Shih, W.W. Liou, A. Shabbir, Z. Yang, and J. Zhu. A New k-ε Eddy Viscosity Model for High Reynolds Number Turbulent Flows-Model Development and Validation. National Aeronautics and Space Administration, Lewis Research Center, Cleveland, Ohio, August 1994.
[5] Marco Lanfrit, Best Practice Guidelines for Handling Automotive External Aerodynamics With Fluent. Fluent Deutscheland GmbH, Birkenweg 14a 64295 Darmstadt/Germany.
[6] W. Seibert, M. Lanfrit, B. Hupertz and L.Kruger, A best Practice for High Resolution Aerodynamic Simulation around a Production Car Shape. 4th MIRA International Vehicle aerodynamics Conference in Warwick, UK, October 16-17, 2002.
[7] Zhu Hui, Yu Hao, Yang Zhigang. Numerical Analysis on Effect Of Back/Front windshield and Hood angle on Atomotive Aerodynamic Drag. State Key laboratory of Advanced Design and manufacturing for vehicle body.
[8] Joel Guerrero. Introduction to Computational Fluid Dynamics: Governing Equations, Turbulence Modeling Introduction and Finite Volume Discretization Basics. State Key laboratory of Advanced Design and manufacturing for vehicle body.
[9] Wilcox, David C Turbulence Modeling for CFD.2nd edition. Anaheim: DCW Industries, pp.174. 1998.
[10] Reynolds, Osborne. On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion. Philosophical Transactions of the Royal Society of London. A, v. 186, pp. 123-164. 1895.

[11] Autozine,”http://www.autozine.org/Archive/BMW/new/3er_F30.html “, published on 23 Jan 2012