Investigation of Aerodynamic Forces on Vehicle using CFD Technique

Deepak B. Kushwaha[1], Vikas V. Chaurasiya[2], Mohd. Raees[3]
[1]*Deepak B. Kushwaha, Theem College of Engineering*
[2]*Vikas V. Chaurasiya, Theem College of Engineering*
[3]*Mohd. Raees, Theem College of Engineering*

***Abstract*— The customer needs are increasing day by day due to which the advance technology plays an important role in the automobile industry. At considerable speed, forces like drag & lift, weight, side forces and thrust acts on a vehicle which significantly affect the fuel consumption. The drag force is produced by relative motion between air and vehicle and about 60% of total drag is produced at the rear end. Reduction of drag force improves the fuel utilization. The drag force can be calculated by Experimental or Computational fluid dynamics (CFD) technique. Due to competitiveness in the market as well as high cost, time and other limitation. It can be advantageous to use CFD over traditional experimental-based analysis. The purpose of this work is to use CFD software for finding out drag, lift forces and other flow features at different velocities on Audi car profile. The velocities are 80, 100, 120 kmph. This study proposes an effective validation of results between experimental method and CFD methods. The software used in this work are Solidworks 2015 (Modelling), GAMBIT 2.4.6 (Meshing), ANSYS Fluent 17.0 (CFD Solver) as well as Tecplot 360 (Post Processing).**

***Index Terms*— Aerodynamics, Ansys-fluent 17.0, CFD, GAMBIT 2.4.2, Solidworks 2015, Tecplot 360.**

I. INTRODUCTION

Now-a-days, the demand of high speed cars is increasing in which vehicles' stability, low fuel consumption, less pollution, are of major concern. When a vehicle is moving on road, the fuel consumption of the vehicle is affected due to aerodynamic forces like drag, lift, weight, side forces and thrust [7]. Aerodynamic drag is the result of interaction between the vehicle shell and the surrounding air molecules. It is caused by relative motion between the air and vehicle which results in a net force opposing motion [6]. Aerodynamic drag increases the square of velocity [8]. Therefore, it becomes critically important at higher speed. Thus, reducing the drag coefficient in an automobile improves the performance of vehicles as it pertains to speed and fuel efficiency [10].

In automotive industry, mainly wind tunnel and computational fluid dynamic approach are used to estimate the drag. Due to competitiveness of the market as well as high cost, time and other limitation, the use of CFD over traditional experimental-based analyses can be advantaged. Since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced practically with no added expenses. CFD has increasingly provided the methodology behind an important design tool for the automotive industry [1].

II. AERODYNAMICS

A. Aerodynamics

"Aerodynamics" is a branch of fluid dynamics concerned with studying the motion of air particularly when it interacts with a moving object [9]. Anything that moves through air is affected by aerodynamics. The rules of aerodynamics explain how an airplane is able to fly.

B. Automotive aerodynamics

Automotive Aerodynamics is the study of the aerodynamics of road vehicles. Its main goals are reducing drag and wind noise, minimizing noise emission and preventing undesired lift forces and other causes of aerodynamic instability at high speeds [11].

C. Factors contributing to flow field around vehicle

The frictional force of aerodynamic drag increases significantly with vehicle speed [2]. The major factors, which affect the flow field around the vehicle, are the boundary layers, separation of flow field, friction drag and lastly the pressure drag.

III. COMPUTATIONAL FLUID DYNAMICS (CFD)

According to Oleg Zikanov [3] CFD can be defined as: "CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer."

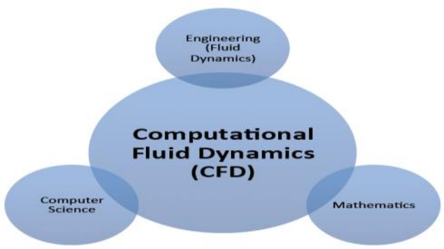

Fig. 1. The different disciplines contained within computational fluid dynamics.

According to this definition, CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (heat and mass transfer). In retrospect, it is integrating not only the disciplines of fluid mechanics with mathematics but also with computer science as illustrated in **Fig. 1**. The physical characteristics of the fluid motion can usually be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD. Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu [4] has discussed how to solve mathematical equations with using CFD.

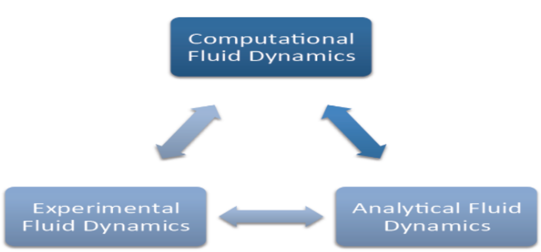

Fig. 2. The three basic approach to solve problems in fluid dynamics and heat transfer

CFD has also become one of the three basic methods or approaches that can be employed to solve problems in fluid dynamics and heat transfer. As demonstrated in **Fig. 2**, each approach is strongly interlinked and does not lie in isolation.

A. How does a CFD code works?

CFD codes are structured around the numerical algorithms that can be engage in fluid problems. In order to provide easy access to their solving power, all commercial CFD packages include difficult user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements.

- Pre-Processor
- Solver
- Post-processor

B. Pre-processor

A pre-processor is used to define the geometry for the computational domain of interest and generate the mesh of control volumes (for calculations). Generally, the finer the mesh in the areas of large changes, the more accurate the solution. Fineness of the grid also determines the computer hardware and calculation time needed [5].

C. Solver

The solver makes the calculations using a numerical solution technique, which can use finite difference, finite element, or spectral methods. Most CFD codes use finite volumes, which is a special finite difference method. First the fluid flow equations are integrated over the control volumes (resulting in the exact conservation of relevant properties for each finite volume), then these integral equations are discretized (producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations [5].

D. Post-Processor

The post-processor provides for visualization of the results, and includes the capability to display the geometry/mesh, create vector, contour, and 2D and 3D surface plots. Particles can be tracked throughout a simulation, and the model can be manipulated (i.e. changed by scaling, rotating, etc.), and all in full colour animated graphics [5].

E. Problem solving with CFD

There are many decisions to be made before setting up the problem in the CFD code. Some of the decisions to be made can include: whether the problem should be 2D or 3D, which type of boundary conditions to use, whether or not to calculate pressure/temperature variations based on the air flow density, which turbulence model to use, etc. The assumptions made should be reduced to a level as simple as possible, yet still retaining the most important features of the problem to be solved in order to reach an accurate solution.

After the above decisions are made, the geometry and mesh can be created. The grid should be made as fine as required to make the simulation 'grid independent'.

IV. GEOMETRY AND DIMENSIONS

Geometric models were modeled using Solidworks 2015 modeling software. For the present analysis only 2-D profile of Audi R8 car model was used. The modeling process involved importing the vehicle blueprints into Solidworks with the help of which, 3D curves were projected. These curves then acted as boundaries to generate surfaces. The final surface model was converted into IGS file format before importing it to Ansys. The **Fig. 3** shows 2D profile of Audi R8 car model and the **Fig. 4** shows the final surface model of the car.

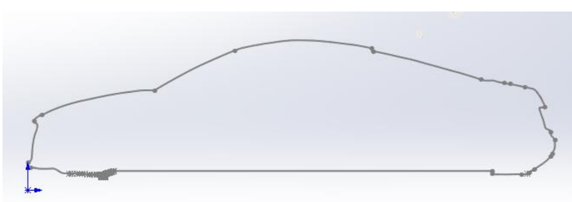

Fig. 3. CAD Profile.

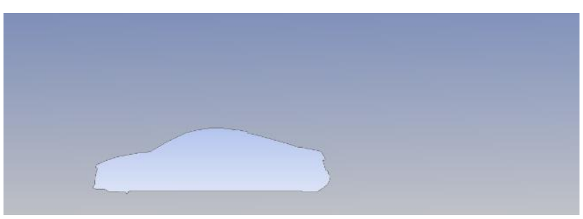

Fig. 4. Surface geometry

V. CREATING FLUID ENCLOUSER

In order to simulate the air flow around the vehicle, a fluid volume needs to be created which will encompass the vehicle. This was done by creating an enclosure around the vehicle and subtracting the vehicle body. This enclosure acts as the air domain.

To reduce the overall computational cost and time, the vehicle was considered symmetric laterally. Dimensions of analysis domain are presented in **Fig.5**. Where L = 4431 mm.

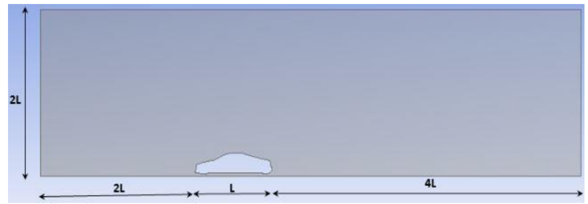

Fig. 5. CFD domain

IV. MESH GENERATION

While generating the mesh, sizing functions were used wherever necessary in order to obtain accurate lift/drag parameters. Two bodies of refinements were added to properly capture the flow in the region closest to the vehicle and also capture the flow in the wake since, boundary layer separation has a significant effect on drag. After meshing problem in GAMBIT, the mesh consists of quads and triangular. The total number of elements obtained was 58.102 thousand. **Fig**. **6 & 7** shows the final mesh.

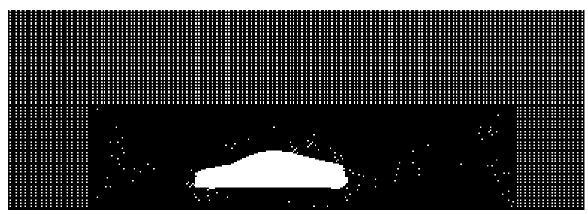

Fig. 6. Mesh away from profile.

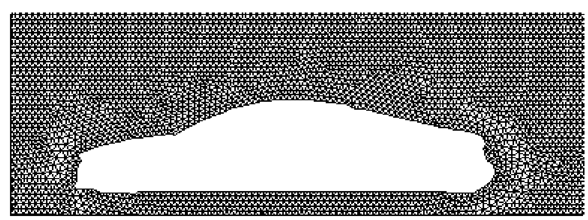

Fig. 7. Mesh closer to profile.

V. BOUNDARY CONDITIONS

Three different velocity of the air at the inlet boundary condition is set in Fluent and velocities are shown in Table I.

TABLE I. INLET VELOCITIES

Sr. No.	Velocities	Unit
1	22.22	m/s
2	27.77	m/s
3	33.33	m/s

The outlet boundary condition is set to pressure outlet with the gauge pressure of 0 Pa. The car contour, the top and the bottom of the virtual wind tunnel are set as symmetry. The density of air is set as 1.225 kg/m3 and the viscosity of air is 1.7894 x 10-5 kg/ms. **Fig. 8** represent the boundary conditions.

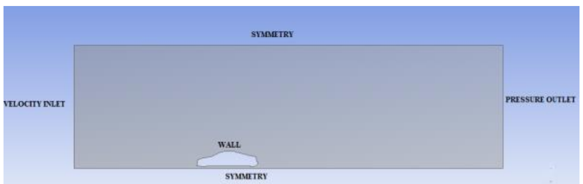

Fig. 8. Boundary conditions

V. SOLVER

For this analysis, a pressure based transient state solver was used. The solution methods, equations used along with the input data are listed below:

- Pressure based transient state solver
- Shear stress transport (SST *k-ω*) model
- Air velocities at inlet are 22.22 m/s, 27.77 m/s and 33.33 m/s respectively

D. Transient flow analysis

Transient flow is the flow, wherein, the flow velocity and pressure are changing with time. When changes occur to a fluid system such as during starting or stopping, in such a situation transient flow conditions exists. Otherwise the system is in steady state. Often, transient flow conditions persist as oscillating pressure and velocity waves for some time after the initial event that caused it.

Time step size (Δt) must be small enough to resolve time-dependent features observed in transient flow and to make sure convergence is reached within the number of Max Iterations per Time Step. The setting selected were listed in Table II.

TABLE II. SOLVER SETTINGS

Sr. no.	Velocities (m/s)	Time step size (s)	No. of time step	Max. iteration/time step
1	22.22	0.057969	14000	20
2	27.77	0.046383	18000	20
3	33.33	0.038646	21000	20

IV. RESULTS

The various flow features and parameter of Audi R8 2D car model are shown in **Fig 9-26**. Table III show the results for C_D with different flow velocities

TABLE III. C_D FOR DIFFERENT VELOCITIES

Sr. No.	Velocities (m/s)	C_D
1	22.22	0.243702
2	27.77	0.241598
3	33.33	0.239977

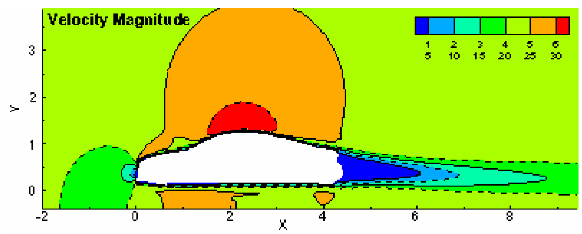

Fig. 9. Velocity contour at 22.22 m/s

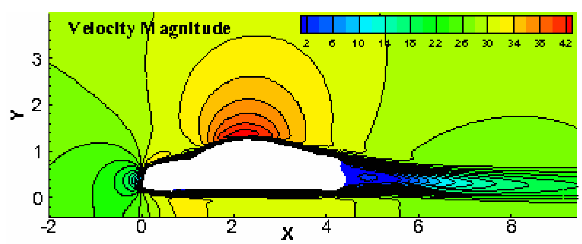

Fig. 10. Velocity contour at 27.77 m/s

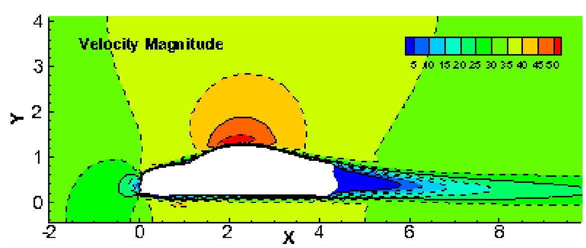

Fig. 11. Velocity contour at 33.33 m/s

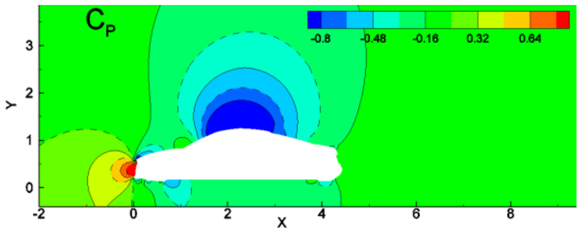

Fig. 12. Coefficient of pressure at 22.22 m/s

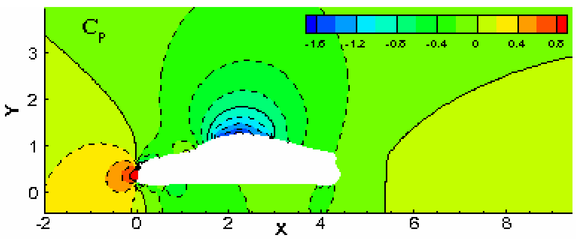

Fig. 13. Coefficient of pressure at 27.77 m/s

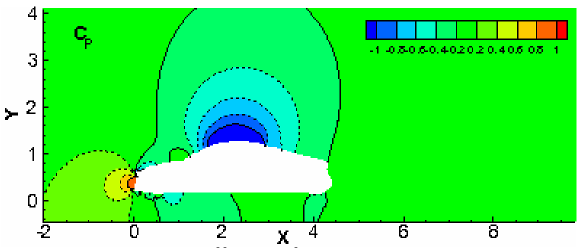

Fig. 14. Coefficient of pressure at 33.33 m/s

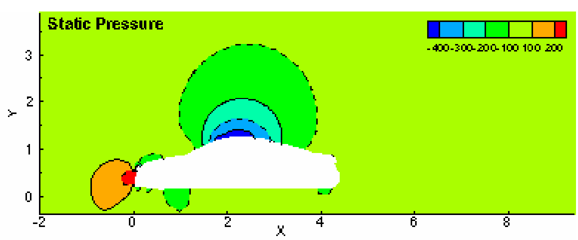

Fig. 15. Static pressure at 22.22 m/s

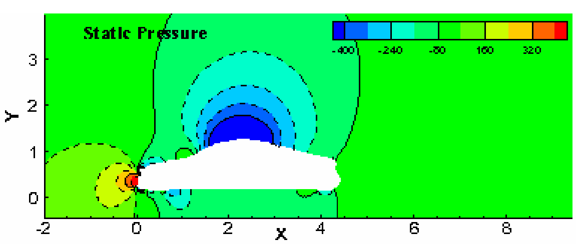

Fig. 16. Static pressure at 27.77 m/s

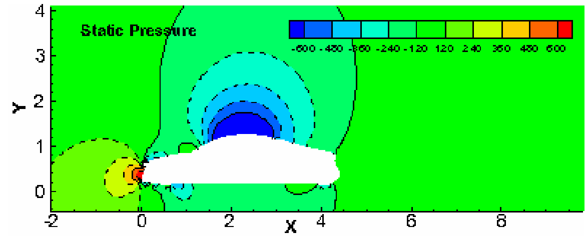

Fig. 17. Static pressure at 33.33 m/s.

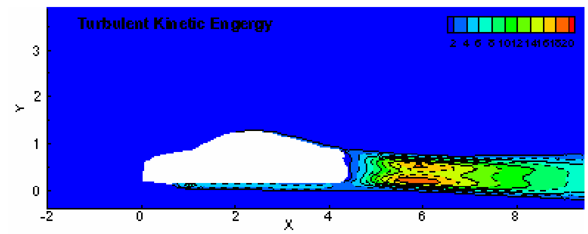

Fig. 18. Turbulent kinetic energy at velocity 22.22 m/s

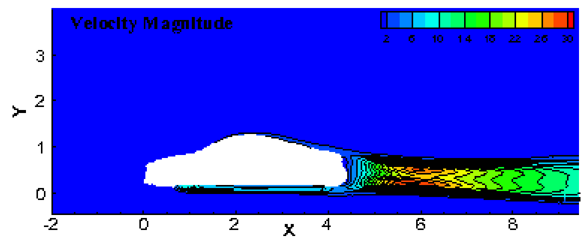

Fig. 19. Turbulent kinetic energy at velocity 27.77 m/s

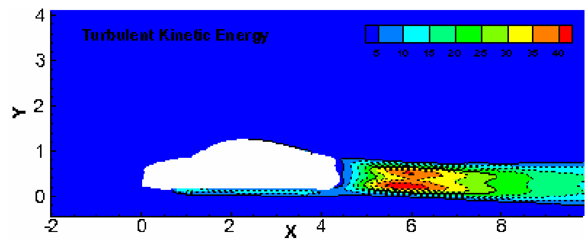

Fig. 20. Turbulent kinetic energy at velocity 33.33 m/s

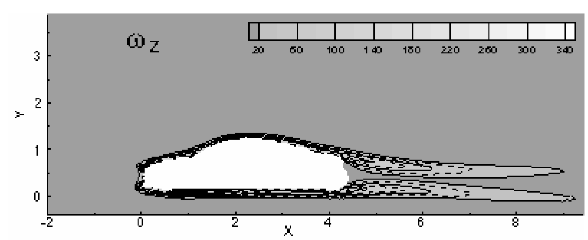

Fig. 21. Vorticity at velocity 22.22 m/s

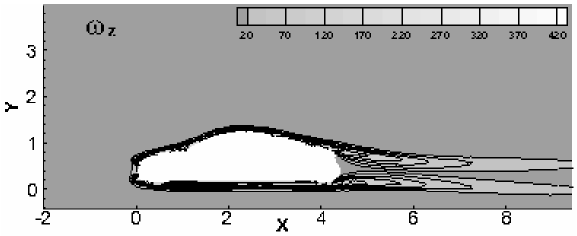

Fig. 22. Vorticity at velocity 27.77 m/s

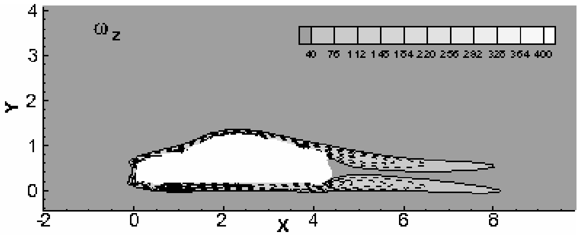

Fig. 23. Vorticity at velocity 33.33 m/s

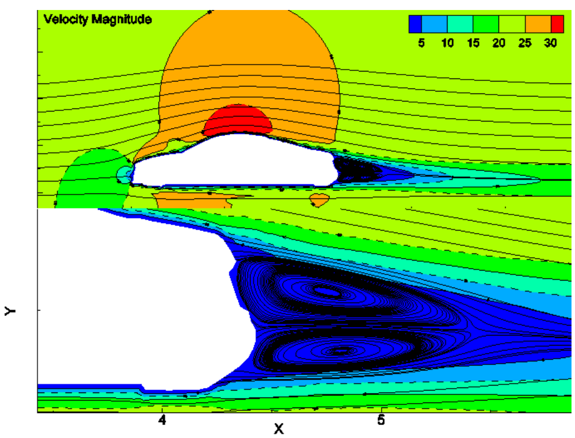

Fig. 24. Streamlines and flow separation at velocity 22.22 m/s

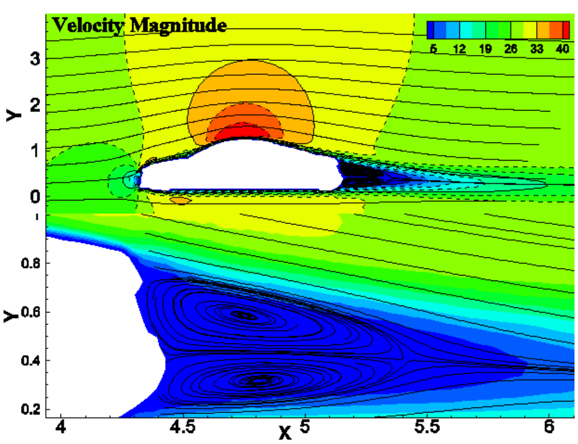

Fig. 25. Streamlines and flow separation at velocity 27.77 m/s

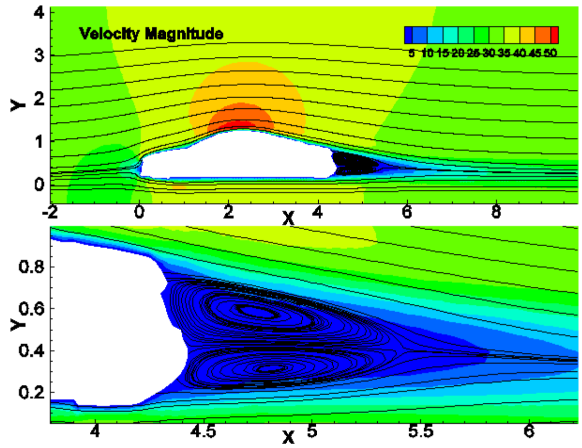

Fig. 26. Streamlines and flow separation at velocity 33.33 m/s

Since 1991 to 2017 the experimental result of coefficient of drag (C_D) of Audi car model ranges from 0.23 to 0.33 [12]. The coefficient of drag (C_D) obtained from current analysis of Audi model using CFD is given in Table III, which is within the range of experimental result. Thus the obtained result is reliable and validate the experimental result.

VII. CONCLUSION

On the basis of obtained result, it can be concluded that CFD can provide near about accurate result in comparison with experimental result. In this work three different flow velocity were studied and it is observed that drag force increases as velocity increases. Thus, require more motive power which affects the fuel consumption. 2D analysis is very helpful and usually preceded by a 3D analysis, because they can provide some basic guidelines that could be used to redesigned and enhance the solution. This approach can significantly shorten the time of analyzing a problem.

ACKNOWLEDGMENT

We express esteemed gratitude sincere thanks to our worthy lecturer guide Prof. Mohd. Raees, our vocabulary does not have suitable words benefiting to high standard at knowledge and extreme sincerity, deviation and affection with they have regularly encouraged us to put heart and soul in this work.

REFERENCES

[1] N. Ashton, A. Revell And R, Poletto, "Grey-Area Mitigation For The Ahmed Car Body Using Embedded DDES", Progress In Hybrid Rans-Les Modelling, Springer International Publishing Switzerland 2015.

[2] Tuncer Cebeci, Jian P. Shao, Fassi Kafyeke, Eric Laurendeau, Computational Fluid Dynamics for Engineers: From Panel to Navier-Stokes, Springer, 2005, ISBN 3-540-24451-4.

[3] Oleg Zikanov, "Essential Computational Fluid Dynamics", John Wiley & Sons, Inc. Hoboken, New Jersey, March 2010.

[4] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, "Computational Fluid Dynamics: A Practical Approach", Butterworth-Heinemann; 1st edition, Burlington, MA, November 2007.

[5] Versteeg H., Malalasekra W., "An Introduction to Computational Fluid Dynamics: The Finite Volume Method", Second Edition, Pearson Education Limited, Essex, England (2007).

[6] Anderson, John D. Jr., Introduction to Flight.

[7] Thomas D. Gillespie, Fundamentals of Vehicle Dynamics, SAE

[8] Joseph Katz, Automotive Aerodynamics, John Wiley & Sons, Ltd, 2106.

[9] https://www.grc.nasa.gov/WWW/K-12/airplane/bga.html

[10] http://nextbigfuture.com/2009/03/reducing-drag-on-cars-and-trucks-by-15.html

[11] https://en.wikipedia.org/wiki/Automotive_aerodynamics

[12] https://en.wikipedia.org/wiki/Automobile_drag_coefficient