Exploring the Influence of Vehicle Body Designs on Aerodynamic Drag

Chenyu Zhang*
St. Andrew's College, Aurora, Canada
* Corresponding Author Email: chenyuzhang2230@gmail.com

Abstract. What are the major causes of aerodynamic drag in vehicles, and how does it impact the fuel efficiency of a vehicle? The more aerodynamic body will experience less air resistance and friction as it travels across the surface, making it more fuel efficient. Many unique aerodynamic designs have been shown to have a lower drag coefficient that traditional cars, but further research is needed. 2 studies investigate the aerodynamics of vehicles using CFD models. The CFD models verify that most of the aerodynamic drag of a car is influenced by the front of the car, and the windshield. For planes, a smaller, more streamlined shape leads to less shear forces and temperature increases in planes, allowing for greater distances to be travelled. These factors should be implemented for the greatest result in vehicles. Improved developments in streamlining vehicles and creating more innovative body designs will create more fuel-efficient vehicles in the future. Future aerodynamic research may also focus on other factors to aerodynamic drag, such as spoilers or the streamlining vehicles for turbulent forces in multiple directions.

Keywords: Aerodynamics, CFD model, fuel efficient, airflow, vehicle.

1. Introduction

In a vehicle, aerodynamics is one of the biggest influencing factors for speed, fuel efficiency, and stability. The goal of a vehicle is to transport people from one place to another quickly, efficiently, and in a comfortable way. Aerodynamics is the way objects flow through air, and having good aerodynamics allows for better airflow across a vehicle. Better airflow means the air experiences less resistance and friction as it travels across the surface. This leads to less turbulence, less energy used, and more speed, as the air does not slow the vehicle down as much. There is less turbulence due to the smoother airflow, leading to fewer irregularities. It requires less energy to travel when the vehicle doesn't need to overcome as much resistance. Finally, the vehicle can travel faster and accelerate faster since it doesn't need to overcome as much air resistance. For planes, having good aerodynamics also allows to better, sustained flight, as the wings can generate more lift. All these benefits come from aerodynamics, and it is one of the biggest factors to consider when designing a vehicle. Many studies have investigated the aerodynamics of cars, planes, and modifications that would change airflow.

Throughout history, understanding in aerodynamics and turbulent forces has led to progressively better designs. Prior to flight with aircrafts, the only things that were able to sustain flight were hot air ballons. The first-time flight was achieved by a powered aircraft was by the Wright brothers on December 17, 1903, with the plane flying in the air for 12 seconds and travelling 120 feet. 1 minute flight would later be achieved in 1907 by Henri Farman. most common design at the time included rigid wings, a rudder, and elevator to control the yaw and pitch [1].

This would later evolve into the biplane, which also included a cockpit, and a more streamlined tail. This reduced the turbulence and drag. After WW1, countries tried to improve aircrafts. The advancement in the shape of airfoils also improved with the introduction of NACA. By the time WW2 came, most plane designs became narrower at the front and only had 1 pair of wings. Eventually commercial airliners became popular, and they adopted a longer fuselage with the wings in the middle rather than the front [2]. The engines also moved from the front to the sides or the back that went on the wings or the back of the plane. Modern fighter jets have much narrower wings that form a triangle shape, allowing for more lift and speed. This evolution has had aerodynamics as the main factor, and recent designs are starting to incorporate more innovative ideas.

The next section will cover related concepts and current research done on aerodynamic vehicle models. Section 3 discusses mathematical tools used in calculating aerodynamic drag and CFD models. Section 4 discusses the results of 2 experiments and their implications on aerodynamic vehicle designs. Section 6 concludes the paper.

2. Literature Review

The current design for vehicle bodies aims to be fuel efficient, durable, and durable. Durability is achieved through better materials and good engineering in parts like the frame, while fuel efficiency is achieved through less loads and resistance. Lighter vehicle bodies reduce the load, and reducing resistance is achieved by making the vehicle more streamlined.

R. Buchheim et al. shows that aerodynamics has the most significant improvements in fuel economy, noise, and costs is improved aerodynamics. The drag coefficient was also directly proportional to the range of the vehicle. Common aspects that are considered include the roof shape, rear shape, underbody, front end, and styling model for aesthetics. A typical full-size car of 77 inches has a drag coefficient of 0.50, while detail optimization lowers it to 0.42. This method used by Volkswagen in the 1970s optimizes small peripheral details to make them more streamlined to the vehicle. Further optimizations include a curved shape at the front of the vehicle. The limitations is the aesthetics and stylizing of the vehicle, which is important to maintain for commercial vehicles. The angle of the front and windshield are also important for achieving the streamlined design, with 28 to 30 degrees being optimal for cars, yet still have proper vision [3].

The data shows that the general idea for an aerodynamic car is a high rear end, a tapered front end, round edges, and low windshield angle. Although outdated, these ideas still exist in current vehicle designs. One example of traditional car studies is Ahmed car, which is a generic car geometry with a slanted back [4]. Experiments with test angles from 0 degrees to 40 degrees of slant lead to different turbulent behaviors.

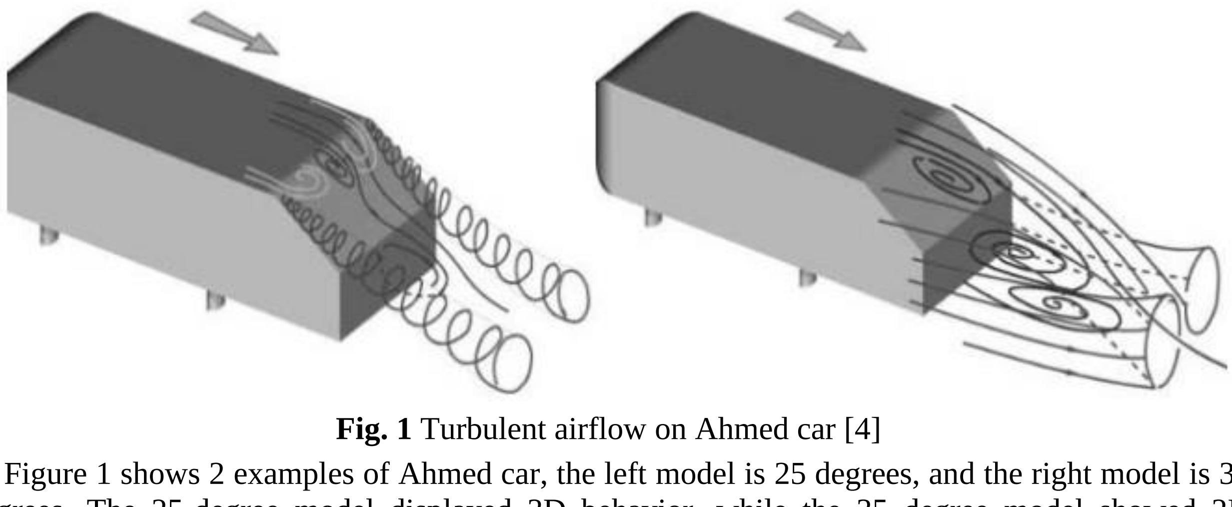


Fig. 1 Turbulent airflow on Ahmed car [4]

Figure 1 shows 2 examples of Ahmed car, the left model is 25 degrees, and the right model is 35 degrees. The 25-degree model displayed 3D behavior, while the 35 degree model showed 2D behavior, with both models displaying turbulent forces (Fig. 1).

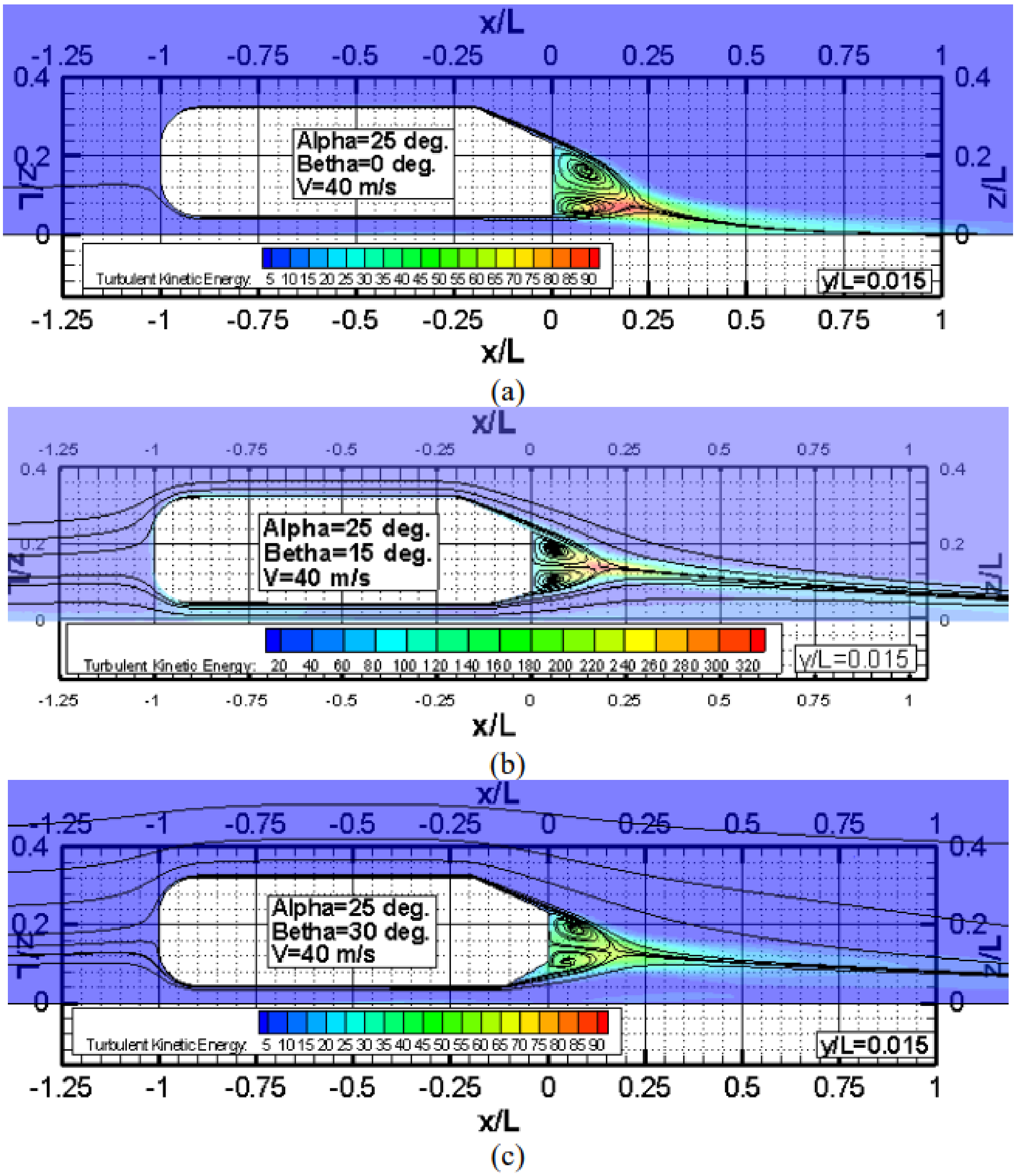


Fig. 2 Turbulent flows off Ahmed car at a) β=0°, b) β=15° and c) β=30° [5]

Research conducted to Seyfettin Bayraktar and Yılmaz Ogun Bilgili model and allow us to better understand the turbulent forces on cars. The results in figure 2 display the turbulent forces at various angles of Ahmed car, giving a better understanding about the optimizations to the back of a car to ensure the lowest turbulent force [5].

Many modern cars also utilize wind tunnels to improve aerodynamic designs, active grill shutters, and active wheel shutters, to minimize the drag caused in those systems [6]. These designs have conceptualized modern aerodynamic bodies for vehicles and are some of the foundations that other car models are built upon.

Better aerodynamics leads to better fuel efficiency, as the vehicle encounters less resistance while in motion. This leads to more energy efficient vehicles that can travel longer and faster. We can see a clear correlation between lower drag coefficient and fuel efficiency. A drag coefficient of cd = 0.32 saves between 10 to 15% more fuel in passenger cars then when the drag coefficient is cd=0.42 [7]. A more aerodynamic body leads to lower fuel expenditure, increasing the fuel efficiency.

Research has been done in to find newer designs that improve aerodynamics. Chowdhury et al created a car design that mimics the design of a boxfish The biomimicry of the boxfish has been shown to have a drag coefficient of 0.28, which is lower than any car on the road. Most drag was caused by vortices at the front of the vehicle, and the rest was very streamlined [8].

Better aerodynamics leads to better fuel efficiency, as the vehicle encounters less resistance while in motion. This leads to more energy efficient vehicles that can travel longer and faster. We can see a clear correlation between lower drag coefficient and fuel efficiency. A drag coefficient of cd = 0.32 saves between 10 to 15% more fuel in passenger cars then when the drag coefficient is cd=0.42 [7]. A more aerodynamic body leads to lower fuel expenditure, increasing the fuel efficiency.

3. Methods and Technical Models

3.1. Fundamental Theories of Aerodynamic Models

Aerodynamic models are used to accurately predict the aerodynamic forces applied to an object, and helps when designing vehicle bodies, since you don't need to create a physical model to see the results.

Bernoulli's Principle is the fundamental theory in fluid mechanics and is used to find aerodynamic drag. It states that an increase in the speed of an incompressible fluid has a relationship with the pressure of the fluid. If the velocity of the fluid increases, the pressure decreases, and as the velocity decreases, the pressure increases. The mathematical equation for Bernoulli's principle is:

$$P + \frac{1}{2}pv^2 + pgh = constant \tag{1}$$

Where: - P is the fluid's static pressure (Pa), - ρ is the fluid's density (kg/m³), - v is the fluid velocity (m/s), - g is the acceleration due to gravity (m/s²), - h is the height of the fluid (m).

Bernoulli's equation is used to explain and calculate the change in air pressure around a vehicle as its velocity changes. As the velocity of a fluid changes around a vehicle, it changes the pressure, leading to different effects. High speed vehicles like cars use this concept to generate downforce, a crucial factor in the stability of a car. Having large changes in pressure around a vehicle contributes to greater aerodynamic drag, slowing vehicles down. Aerodynamic drag is the main source of resistance that any vehicle experiences, and the equation for aerodynamic drag is:

$$PF_d = \frac{1}{2}C_d\rho Av^2 \tag{2}$$

Where: - *Fd* is the drag force (N), - *Cd* is the drag coefficient, - ρ is the air density (kg/m³), - *A* is the frontal area (m²), - *v* is the velocity of the vehicle relative to the air (m/s).

Although Bernoulli's equation takes pressure and velocity into account for a vehicle, viscosity and turbulence are factors that need to be considered in a vehicle. This leads to a deeper analysis with Reynolds Number and Boundary Layers.

Reynold's number (Re) is a dimensionless value that determines the type of flow. It represents a ratio of inertial forces to viscous forces and is used to predict is a flow is laminar or turbulent. The equation for Reynolds number is:

$$Re = \frac{\rho vL}{\mu} \tag{3}$$

Where: - ρ is the fluid's density (kg/m³), - v is the characteristic velocity (m/s), - L is the characteristic length (typically the length of the vehicle or the path of the fluid flow, m), - μ is the dynamic viscosity of the fluid (Pa/s).

A lower Reynold's number (Re) is a laminar flow, while a high Reynold's number means a turbulent flow. A laminar flow flows evenly and parallel to each other, while a turbulent flow experiences a lot of mixing and leads to irregularities. This is crucial for vehicles, as the type of airflow influences the amount of drag that is experienced by the vehicle, and increased turbulent flows in aircrafts lead to increased turbulence. By having a streamlined vehicle shape, the turbulent flow can be reduced, leading to less drag and better fuel efficiency.

The boundary layer is a layer of fluid that surrounds an object, or a bounding surface. This layer of fluid is the fluid that affects and interacts with an object and this layer is the cause of drag and aerodynamics in objects. The boundary layer of a fluid has less effect on the object the farther it is from the object and has a decreased velocity. The state of the boundary layer determines the friction drag that is applied to objects by the air molecules: Laminar or turbulent. The boundary layer thickness (δ) is dependent on the shape of the object and the Reynolds number (Re):

$$R\delta = \frac{5L}{\sqrt{Re}} \tag{4}$$

A laminar flow has a parallel flow with little to no mixing, while a turbulent flow experiences many irregularities. The thickness of the boundary layer increases with distance from the leading edge of the object. Increased velocities lead to thinner laminar boundary layers, which transition to turbulence more easily.

Reducing the boundary layer of a vehicle and the amount of turbulence caused by the boundary layer decreases the number of turbulent forces and improves the aerodynamic performance.

By effectively utilizing Bernoulli’s Principle, Reynolds number, and boundary layer theory, vehicle designs can be significantly optimized for better aerodynamic efficiency, leading to reduced drag and improved fuel efficiency and performance.

3.2. Computational Fluid Dynamics (CFD) Techniques for Vehicle Design

Computation Fluid Dynamics (CFD) is a brand of fluid dynamics that uses math to calculate and predict fluid transfer and how a fluid interacts with another object. It uses equations to predict the airflow or flow of a fluid around a model that you input. It also allows you to visualize the flow and change the different scenarios without a physical product. It is crucial in the design process of vehicles, as it is used to simulate the drag that the vehicle body will produce and any turbulent forces.

CFD works by separating fluid domains into smaller control volumes, where it is solved by the Navier-Stokes equation. It describes the motion of fluid substances, and is quantified by:

$$P\ \left(\frac{\partial v}{\partial t} + v * \nabla v\right) = -\nabla P + \mu \nabla^2 v + f \tag{5}$$

Where: - ρ is the fluid density (kg/m³), - v is the velocity vector (m/s), - P is the pressure (Pa), - μ is the dynamic viscosity (Pa*s), - f is the external body forces (N).

When designing vehicles, CFD can simulate the airflow around objects in different scenarios, such as the formation of turbulent flows around the vehicle. CFD offers many advantages: - Cost-efficiency: CFD reduces the need for expensive physical prototypes. - Flexibility: Multiple design iterations can be tested virtually. - Precision: CFD can capture detailed flow characteristics, including localized pressure distributions and small-scale vortices, enabling precise optimization of vehicle aerodynamics. CFD models help simulate airflow and improve our aerodynamic vehicle designs and our understanding of aerodynamics.

Direct Numerical Simulation (DNS) is the most accurate method for simulating fluid flows as it solves the full Navier-Stokes equations without any turbulence modeling. DNS resolves all scales of motion down to the smallest eddies, thus offering unparalleled accuracy in capturing fluid behavior. The governing equation is:

$$\rho\left(\frac{\partial v}{\partial t} + v * \nabla v\right) = -\nabla P + \mu \nabla^2 v - \nabla * \tau_{SGS} \tag{6}$$

However, DNS is computationally intensive because it requires very fine spatial and temporal resolution, making it impractical for large-scale simulations like entire vehicles. It is primarily used for high-precision studies of small-scale flows, such as investigating flow separation or transitional flows in specific areas like the boundary layer or wake regions.

3.3. Experiments and Model Evaluation

To assess the aerodynamic performance of different vehicle body designs, wind tunnel testing, computational fluid dynamics (CFD) simulations, and real-world road testing are commonly used. Many experiments test the aerodynamics of vehicles over different speeds and model the results using a CFD model. The key metrics for evaluating aerodynamic performance are the **drag coefficient (Cd)**, and the air velocity. These metrics are important, as a lower drag coefficient quantifies indicates better aerodynamic efficiency. Air velocity is also crucial, as it influences the pressure exerted on the vehicle, and the stress. Reducing the drag coefficient directly improves fuel efficiency, especially at higher speeds where air resistance has a greater influence. These key metrics allow for a better evaluation of aerodynamic drag.

Current research on the factors that influence aerodynamics have been significant to providing information what can be improved in designs, as well as verifying the accuracy of CFD models.

The article Assessing On-Road Emission Flow Pattern under Car-Following Induced Turbulence Using Computation Fluid Dynamics (CFD) Numerical Simulation, by Xueqing Shi et al goes over the on-road emission flow patterns to understand and control atmospheric pollution, a growing issue in China.

A CFD simulation was created to accurately predict and model the results of the experiment. The traffic, CO2 concentration, and other measurements were considered. The results of the CFD model are shown in the image below.

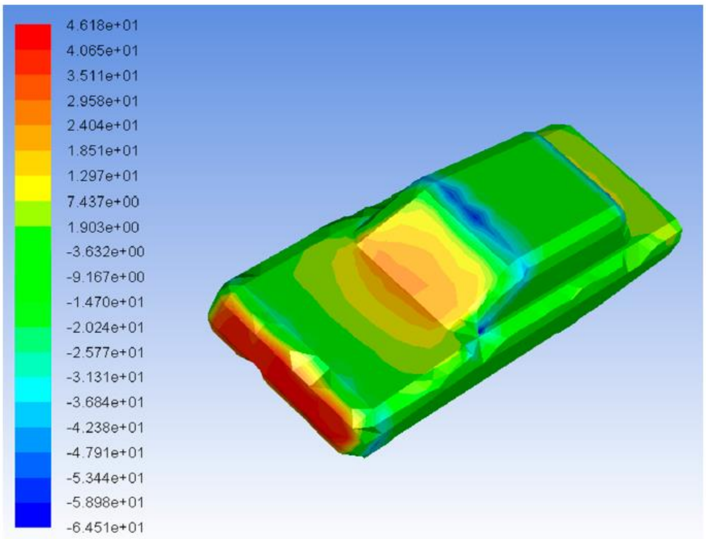
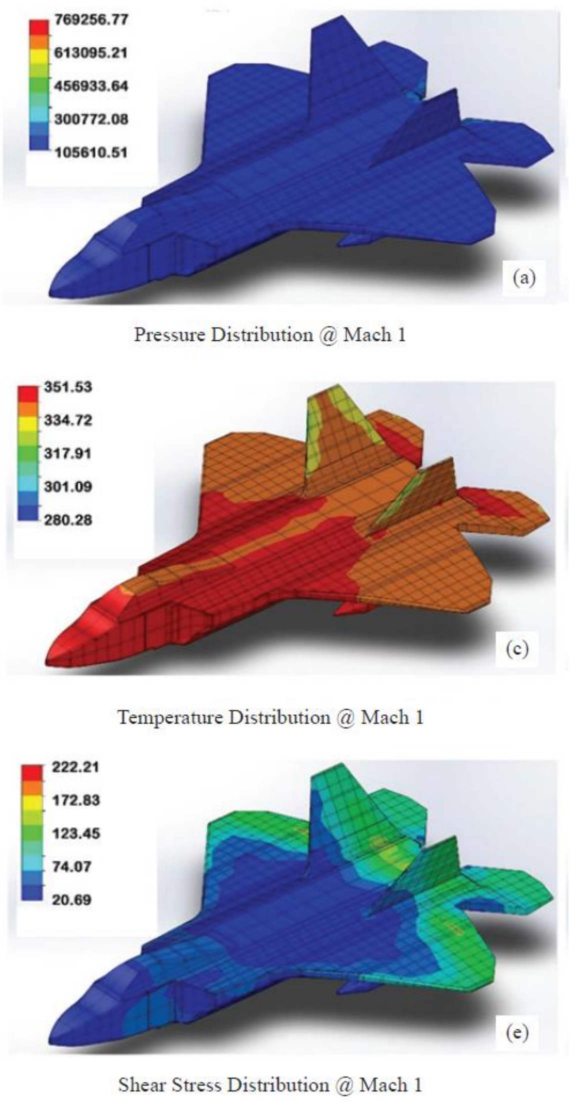


Fig. **3** CFD model of the pressure coefficient of the car [9]

From the image we can see the accuracy of the CFD model, accurately showing where spots on a car have high pressure. For a car, the main spots where there is pressure is the front of the car, and the window, which are less streamlined. This is due to a lower air velocity due to blocking, and higher pressure. To improve aerodynamics and reduce the pressure across a vehicle, streamlining improvements need to be done to the front.

The second study, Comparative Analysis of Aerodynamic Characteristics of F16 and F22 Combat Aircraft using Computational Fluid Dynamics by Lohit Malik and Abhishek Tevatia show the air flow over fighter jets, and the relation it has with their efficiency and performance.

They researched the Lockheed Martin F16 Raptor and General Dynamics F22 Flying Falcon. The F22 comes with twin engines, while the F16 only comes with 1. Both aircrafts were replicated in a SolidWorks model before being put into a CFD model at different speeds and angles. The results at mach 1 for both F22 and F16 are shown below:



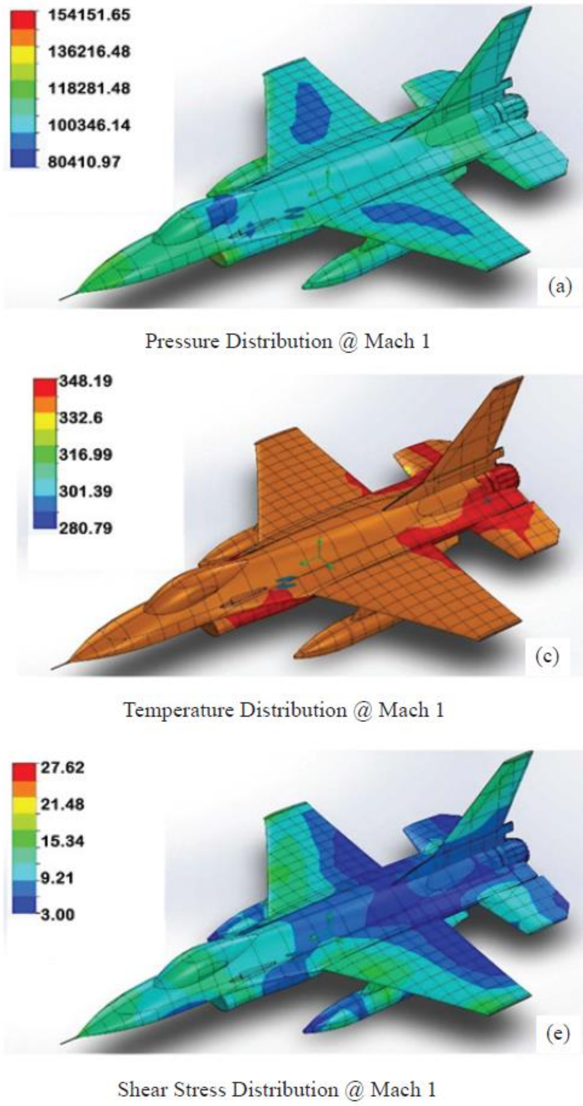


Fig. 4 Stress distribution of an F22 [10]　　**Fig. 5** Stress distribution of an F16 [10]

From Fig, we can see that the F22 has better airflow, and a slightly lower pressure distribution across the entire aircraft. The majority of the shear stress is at the back, where the airflow transitions to laminar to turbulent. The F16 is more streamlined, and has less air friction, as the shear stress distribution is <21, while the F22 has spots where the shear stress is >120. The temperature is also much lower on the F16, with the hottest spot being the location of the engine. This is in contrast to the F22, where the less streamlined design and greater air friction leads to higher temperatures at the front of the aircraft. The improved design for the F16 allows for greater performance, as it can handle greater temperatures for a longer time, and experience less stress, allowing it to be used for longer periods of time.

Both studies show that the most important aspect of an aerodynamic design is the front of the vehicle, as it influences the airflow, and is subject to the most stress.

Current CFD models are accurate and can be used to measure the aerodynamics and forced applied to many conventional vehicles. Experimentation with these models have also provided key information on factors that influence vehicle aerodynamics the most, and what changes can be made. By verifying the accuracy of CFD models, they can be used to further validate and research non-traditional models. Future experimentation and improvements in our knowledge of aerodynamics will allow for better, more efficient vehicles in the future.

4. Conclusion

The purpose of the studies above is to examine our current understanding the most important aspects to the aerodynamics and fuel efficiency of a vehicle, the application of CFD models. Xueqing Shi et al highlights the drag forces in cars while driving. With the front being less streamlined, much of the aerodynamic drag, and the area with the most pressure is the front of the car and the windshield. Lohit Malik and Abhishek Tevatia compare the aerodynamic drag forces of F22 and F16 jets, and how the difference in design influences the drag forces and temperature. The F22 is shown to be less streamlined and endure greater shear forces and temperature. The improved design of the F16 at the front allows for less forces, and makes the vehicle more efficient in the air, enduring longer periods of use. Both studies show that the front of a vehicle has the greatest influence on the aerodynamic drag, and making vehicles more fuel efficient will require more streamlined front ends. Both studies show CFD models with the airflow coming directly from the front and doesn't consider airflow coming in different directions of the vehicles (e.g. a gust of winds against the side of a car). Improvements in CFD models may allow us to calculate airflow coming from multiple directions in the future and improve our understanding in that field. Future studies for cars can look at and analyses or research streamlining bodies to counteract turbulent forces, as well as improve windshield aerodynamics without reducing vision. Future jet designs should focus on a narrower front, and future research should be conducted on the shear forces caused by aerodynamics, and how it impacts the longevity of planes, but also their fuel efficiency. Future research and development in this field will lead to new breakthroughs that change and improve vehicle efficiency.

References

[1] Wegener PP. What makes airplanes fly?: History, science, and applications of aerodynamics. Springer Science & Business Media; 1997.

[2] Anderson JD, Anderson JD Jr. A history of aerodynamics: And Its Impact on Flying Machines. Cambridge, England: Cambridge University Press; 1998.

[3] Buchheim R, Deutenbach K-R, Lückoff H-J. Necessity and premises for reducing the aerodynamic drag of future passenger cars. SAE Trans J Mater Manuf [Internet]. 1981; 90:758–71. Available from: http://www.jstor.org/stable/44631713.

[4] Kuntz M, Menter FR. Ahmed car. In: Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM). Springer Berlin Heidelberg; 2006. p. 335–46.

[5] Bayraktar S, Bilgili YO. Effects of under body diffuser on the aerodynamics of a generic car [Internet]. Org.tr. [cited 2024 Sep 22]. Available from: https://dergipark.org.tr/en/download/article-file/533658.

[6] Al TT et. Simulation and Optimization of an Aluminum-Intensive Body-on-Frame Vehicle for Improved Fuel Economy and Enhanced Crashworthiness - Front Impacts [Internet]. Jstor.org. [cited 2024 Aug 18]. Available from: https://www.jstor.org/stable/26268780?workspaceFolderId=9e6cdcd2-9077-4605-9d4a-acd9d48a6795&orderBy=updatedOn&orderType=desc&index=3.

[7] Janssen and H.-J. Emmelmann L. Aerodynamic Improvements - A Great Potential for Better Fuel Economy [Internet]. Jstor.org. [cited 2024 Aug 30]. Available from: https://www.jstor.org/stable/44611098?workspaceFolderId=9e6cdcd2-9077-4605-9d4a-acd9d48a6795&orderBy=updatedOn&orderType=desc&index=2&seq=3.

[8] Chowdhury H, Islam R, Hussein M, Zaid M, Loganathan B, Alam F. Design of an energy efficient car by biomimicry of a boxfish. Energy Procedia [Internet]. 2019; 160:40–4. Available from: https://linkinghub.elsevier.com/retrieve/pii/S1876610219312007.

[9] Shi X, Sun D (jian), Fu S, Zhao Z, Liu J. Assessing on-road emission flow pattern under car-following induced turbulence using Computational Fluid Dynamics (CFD) numerical simulation. Sustainability [Internet]. 2019 [cited 2024 Sep 12];11(23):6705. Available from: https://www.mdpi.com/2071-1050/11/23/6705.

[10] Tevatia LMA. Comparative Analysis of Aerodynamic Characteristics of F16 and F22 Combat Aircraft using Computational Fluid Dynamics. 2021 Mar.