THE DESIGN AND CONSTRUCTION OF A FUEL-EFFICIENT URBAN-CONCEPT VEHICLE

BY

ADEROGBA, TIMOTHY AYOOLUWA (MEE/2012/004)

OLUSOLA, EMMANUEL OLUWASEGUN (MEE/2012/044)

A PROJECT SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE AWARD OF BACHELOR OF SCIENCE DEGREE IN MECHANICAL ENGINEERING

OBAFEMI AWOLOWO UNIVERSITY ILE-IFE, NIGERIA.

MAY, 2018

CERTIFICATION

This is to certify that ADEROGBA Timothy Ayooluwa (MEE/2012/004) and OLUSOLA Emmanuel Oluwasegun (MEE/2012/044) submitted this project on the Design and Construction of a Fuel-Efficient Urban Concept Vehicle in partial fulfilment of the requirements for the award of Bachelor of Science Degree in Mechanical Engineering, Obafemi Awolowo University Ile-Ife, Nigeria.

Dr. A.O. Muritala

(Supervisor)

Signature ....

Date………16/05/2018…….

Dr. D. A. Adetan

(Head of Department)

Signature ....

Date…………………………….

DEDICATION

This work is dedicated to the Almighty God.

ACKNOWLEDGEMENTS

Firstly, our appreciation goes to God Almighty for seeing us thus far and for His unending support throughout the course of this project.

Our appreciation also goes to our supervisor, Dr. Muritala for his counsel and encouragement.

We will like to appreciate our parents, Pastor E.O. Dedeigbo, Mrs. E.A. Olusola and Mr. and Mrs. Aderogba for their support, morally and financially.

TABLE OF CONTENTS

ABSTRACT

The rapid increase in the price of refine petroleum products motivates various researches on the design of fuel efficient vehicles. A very important design consideration that greatly affects the fuel consumption of vehicles is the aerodynamic characteristics, which are made up of drag, lift and pressure coefficients. This project enumerated the designs of fuel efficient urban concept vehicles and thereafter conducted numerical simulation of flow around the vehicle prototype aiming at analyzing the aerodynamic behavior at different velocities. The design of the car body was carried out to meet the rules and standards of the Eco-Marathon competition organized by Shell. The numerical method involves pre-processing, processing and post-processing activities that was accomplished using ANSYS FLUENT software. The models were developed using Autodesk Inventor and was exported to Fluent where the numerical computation was carried out. The results of the simulation were obtained inform of pressure and velocity contour, drag and lift coefficients, and pressure coefficient. Model 6 has the lowest drag coefficient and is chosen as the best model for the fuel efficient urban concept vehicle.

LIST OF FIGURES

LIST OF TABLES

CHAPTER ONE

INTRODUCTION

1.1 Background

The world's ever-increasing populace needs energy to carry on work and production. Our modern life is built on energy. We use it to heat, to cool, to light up our cities. It powers our cars and flies our planes. It lights our stores and keeps us switched on and plugged in.

Fossil fuels such as petrol are dominating when it comes to considering sources of energy; especially for transportation purposes. Advanced technologies such as biofuels and electric vehicles are taking off in the world, but will require high technological advancement, major financial investments and national policy support in order to scale up. Thus, they are not feasible for a short-term sustainable goal in Nigeria. For the meantime, oil fuel remains the dominant fuel for our transport system.

The world can expect energy prices to continue their generally upward spiral in the years ahead if global energy policies remain the same, the current consumption rates will cause world energy demand to increase 1.6 per year until 2030. (IEA, 2017[1])

Automobile producers provide private goods (vehicles) for private profit (investors), but externalities (emissions) are generated with costs that are publicly shared. For example, costs associated with driving high-emission vehicles in the southern coast of California can generate pollution costs estimated at $10,000 or more per year. (Michalek et al, 2004)

[1] www.worldwatch.org/node/5936

Meanwhile, beyond the monetary pricing of oil are other expensive costs such as the environmental emissions leading to global climate change. To solve these problems, we have to reduce the use of fuel. This brings about the adventure to search for fuel-efficient technologies in various energy applications including automobiles.

1.2 Problem Statement

One of the main issues our society is facing is the constant increase in the temperature of the earth's atmosphere, also known as global warming. A rise in the temperature of the atmosphere can cause ice to melt around the Earth's poles, a rise in sea level, and an increase in rainfall and snowfall worldwide. Greenhouse gases produced by the burning of fossil fuels mainly cause this phenomenon. There is a high level of fuel consumption in automobiles; leading to expensive running cost, large amount of emissions etc. Prevalent rise in energy prices and rise in global emissions is a call for Engineers to find a sustainable, environmental-friendly and conservative energy solutions. Out of so many solutions embedded in the switch to harnessing Green Energy like renewables and so on, having weighed the different options available, considering the Nigerian major source of energy, which happens to be fossil fuel, which has come to stay even for the next two decade. This consciousness have then made us realize that there's an immediate necessity to develop a fuel efficient gasoline-powered vehicle that is economic, environmental and ecologically friendly which will serve as a model for sustainable transportation, thereby solving both the immediate and eventual problems engulfed in the energy sector of production and conservation.

This project therefore pushes forth to solve the challenge of minimizing and conserving energy consumption in vehicles.

1.3 Objectives of the Study

The study aims at satisfying the following general and specific objectives as highlighted subsequently:

General Objective – Pursuit of Energy Efficiency:

The overall aim of this project is to design, build and develop an urban-concept vehicle that would be fuel-efficient.

Specific Objectives:

- To design and simulate a concept vehicle that would have lesser drag coefficient than a conventional car.
- To construct a scaled model of the body shell for the concept car using lighter weight materials.

1.4 Significance of the Study

The concern of the environment has recently led to regulations on fuel consumptions and exhaust emissions. Fuel consumption is particularly important as it represents the technology of the automobile industry. Diverse innovations have been developed by many automotive manufacturers and promise high engine efficiency ratings, low aerodynamic drag coefficient and eventually, significant fuel economy. This study can give many opportunities including development of new designs and innovations for urban road use.

This study may also serve as a future reference for individuals who are interested in participating in Shell Eco-marathon competition or anyone who would be engaged in such a study. Finally, this may also be an innovation for designing efficient and effective aerodynamic design for vehicles.

1.5 Scope and Limitations of the Study

For the vehicle design, dimensions and safety appendages, the study considered the Shell Eco-marathon Europe 2016 official rules for Urban-concept vehicles, which will be highlighted in the methodology.

The Shell Eco-Marathon is an international student competition organized by the Shell Company; challenging students to design, build and race energy-efficient cars. The competition requires entry designs to focus on maximizing vehicle's mileage rather than attaining high speed. The shell Eco-marathon therefore gives award to the cars that go the farthest distance on one liter of fuel.

The design, simulation and fabrication of the concept car developed in this thesis is mainly concentrated at improving aerodynamics for fuel efficiency. This is carried out through the CFD study of the airflow around prospective car models from which the most aerodynamically efficient design will be chosen.

CHAPTER TWO

LITERATURE REVIEW

2.1 Energy in Nigeria

Several energy resources are available in Nigeria in immense proportions. This includes conventional and non-conventional (renewable) energy resources, and they are vastly distributed across the regions of the country. The energy resources in Nigeria are classified as conventional and renewable energy resources, and they are discussed as follows:

2.1.1 Conventional Energy Resources

Nigeria has considerable reserves of conventional energy resources. It is one of the world's largest producers of oil and it has the largest reserves of natural gas in the African Continent. It therefore became the world's fourth leading exporter of liquefied natural gas (LNG) in 2012. Nigeria is also a member of the Organization of the Petroleum Exporting Countries (OPEC), which it joined in 1971 after over 10 years of oil production that began in the late 1950s. Coal reserves stand at 2.175 billion tons, but production has long since ceased (in the 1950s) as the government has concentrated on the oil and gas resources. (Emodi, 2016)

2.1.1.1 Crude Oil

Nigeria produces mostly light sweet crude oils that are predominantly exported to the world market. In Nigeria, commercial production of crude oil began in 1958 based on proven recoverable reserves of 1.48×10^6 billion tons. Production rose from an initial quantity of 3.1 million metric tons to 20.3 million tons in 1960, 54.2 million tons by 1970, and 104.1 million tons in 1980, all in response to demand from international markets rather than from domestic demand. On the average, local consumption accounted for just 3 % of production, while the remaining 97 % was exported. Since 1980, three domestic petroleum refineries have supplied petroleum products for local consumption: The Kaduna Refinery with a capacity of 110,000 bbl/d (barrels per day), Port Harcourt Refinery with a capacity of 210,000 bbl/d, and Warri Refinery with a capacity of 125,000 bbl/d (Oyedepo 2014).

The production of crude oil in Nigeria increased rapidly between 1980 and 2012; however, the rate of increase was dependent on the economic and geopolitical situations in both producing and consuming countries. Nigeria's current production capacity of 2.4 million bbl/d remains low because of problems in the Niger Delta and OPEC production restrictions. However, projections have placed future (2030) production at over 5.0 million bbl/d (Ajao et al. 2009).

2.1.1.2 Coal

Coal was the first conventional fuel to be discovered and used in Nigeria. It was discovered in Enugu State, in the south-eastern region of Nigeria in 1909. The first coal mine, the Ogbete drift mine, was opened in 1916 with an output of 24,500 t. Its operation was merged with that of others within the country in 1950 following the formation of a new corporation known as the Nigerian Coal Corporation (NCC). The responsibility of the NCC in holding a monopoly on coal production (including coke mining) and sales was to exploit coal resources. The Polish firm KOPEX was in charge of its management from the NCC's formation until its collapse after the Nigerian Civil War in 1970 (Nwaobi et al. 2005).

Following the discovery of coal in commercial quantities (1916–1980), the total cumulative production was about 25.3 million metric tons. By 1980, oil contributed 70 % to electricity generation, compared with 25 % and just 1 % for gas and coal, respectively. In terms of coal consumption, the Nigerian Railway Corporation, NEPA, and cement companies consumed about 95 % of the coal produced in Nigeria for heating cements. Coal consumption by the Nigerian Railway Corporation fell to 60 % in 1958, less than 30 % in 1966, and to an insignificant level by 1986. NEPA's coal consumption fell from 30 % in 1966 to an insignificant level in 1970, and the last remaining coal plant was shut down in 1992 (Enibe and Odukwe 1990).

As a result of the loss of its largest consumers, the NCC began to export its desirable low-sulfur content coal to the United Kingdom and Italy. In 1999, the NCC lost its monopoly over the Nigerian Coal Industry (NCI) following the implementation of the federal government's privatization policy by the then President of Nigeria (i.e. President Olusegun Obasanjo). In 2002, work stopped at all NCCoperated mines and the government established a technical advisory committee to revive the NCI (Odesola et al. 2013a, b).

Current coal reserves in Nigeria are estimated to be 2.75 billion metric tons and the nation's proven reserves stand at 639 million tons. The locations of the coal deposits in Nigeria are mostly in the eastern parts of the country. However, the coal reserves have not been fully explored or even marginally developed despite the long history of the coal industry. They are mostly lignite and sub-bituminous, although some are high-volatility bituminous deposits (Akubo et al, 2013).

2.1.1.3 Natural Gas

The estimated proven reserves of natural gas in Nigeria stand at 182 trillion cubic feet (TCF) with a mean gauge pressure of about 12 bar, a calorific value of 35 mJ/m^3, and a mean specific volume of 1.56×10^{-3} m^3/kg. In 2012, the production rate was about 1.35 TCF of dry natural gas, making Nigeria the 25th largest producer of dry natural gas in the world (ECN, 2007).

Natural gas reserves are located in the Niger Delta region of Nigeria (South South). In the past, Nigeria flared about 73 % of its gas because of poor infrastructure, which placed Nigeria second in the list of gas-flaring countries. However, because of the efforts of the Nigerian government to reduce gas flaring through the financing and provision of relevant infrastructure to use the previously flared gas, Nigeria is now 365th on the list (Ibitoye, 2014).

2.1.2 Non-conventional Energy Resources (Renewable Energy)

Renewable energy plays a vital role in meeting the needs of both rural and urban areas of the country in terms of sustainable development (Hui 1997). The development and proper use of renewable energy should be given high priority, especially now that the issues of climate change and global warming are among the most critical issues discussed by the various governments of the world. Developed and developing countries are now adopting renewable resources in order to achieve energy sustainability (Oyedepo, 2012).

Nigeria is blessed with an abundance of renewable energy resources that must be fully harnessed, developed, and properly used. However, the development of renewable energy has so far been slow, and the desperate situation of the energy sector in Nigeria can only be resolved if adequate policies are implemented to attract investors in renewable energy to Nigeria. (Emodi, 2016). Renewable energies at Nigeria's disposal include Biomass, Biogas, Hydropower, Solar, and Wind.

Despite the large energy resources in Nigeria, energy consumption is relatively low compared with other African countries with comparable energy resources. This low energy consumption is due to the recurrent scarcity of petroleum products at vehicle petrol stations, while frequent electricity "black-outs" have resulted in a high reliance by the Nigerian populace on personal electricity generators. (Emodi, 2016)

Despite the scarcity of petroleum products, energy demand has been increasing in Nigeria, because of the increase in economic development and the population growth. According to Sambo et al. (2006), the major driver behind increasing energy demand is the population growth, while the most important determinant is the level of economic activity, measured by the country's gross domestic product (GDP).

2.2 Fuel Economy in Transportation

Song et al. (2010) describe fuel economy as the ratio of the driving distance to the fuel consumption in a whole cycle. The performance of a car engine can be described in terms of good fuel economy, low emissions, high-power to weight ratio, onboard energy and good drivability. (Adeniyi 2008; Turrentine and Kurani 2007).

Since 1939, Shell researchers began experimenting on vehicles that could make more mileage per gallon (mpg) of fuel and now annually, since 1985, young engineers and scientist have been challenged to develop cars that can make extreme mileage on a gallon (Shell, 2012).

Internationally, transportation is responsible for 23% of greenhouse gas emissions, but this value is expected to rise as cars become economically viable for millions of citizens of developing countries. Therefore, a reduction in automobile carbon emissions would significantly impact national and global greenhouse gas emissions and is certainly an important part of the management of global climate change.

Automobile fuel efficiency is a good place to begin the effort to limit greenhouse gas emissions because several different policy objectives independent of climate change push for decreased consumption of oil. In particular, politicians concerned about the national security implications of the massive importation of Middle Eastern oil, economists concerned about the importation's effect on the current account deficit, and public health experts concerned about the effect of automobile exhaust on cancer rates and respiratory disease all recognize harms in the nation's consumption of oil (Collina, 2005).

Examining the recent past and projecting forward, global transportation consumed about 43 million barrels of fuel per day in 2005, or about 52 percent of the total liquid fuels consumed. As shown in Figure 2.1, by 2030 the share of liquid fuels estimated to be consumed for transportation is expected to rise to 60 percent, while the share consumed by others sectors is expected to fall marginally (EIA, 2011).

Considering the amount of energy used in the transportation sector by mode or vehicle type, Figure 2.2 shows that fuel consumption by all modes is growing, and that light-duty vehicles (LVD) and trucks combined consume 66 percent of energy used for transportation (average share over the 30-year period 2000 to 2030) (ICCT, 2014).

The efficient regular kinds of vehicles, as a comparison, have typical mileage of about 50 mpg, which is about 21 km/litre. According to Earlewine et al. (2010), 783 vehicles were surveyed in real time using Global Positioning System (GPS) tracking technique and found the economy to be in the range 12.5 – 66.7 km/litre.

Finally, fuel efficiency is one of the few areas in the climate change debate where the government has a history of regulation, which it can rely upon to legitimize its call for fuel efficiency and utilize as a mechanism of change. (Steiner et al, 2006)

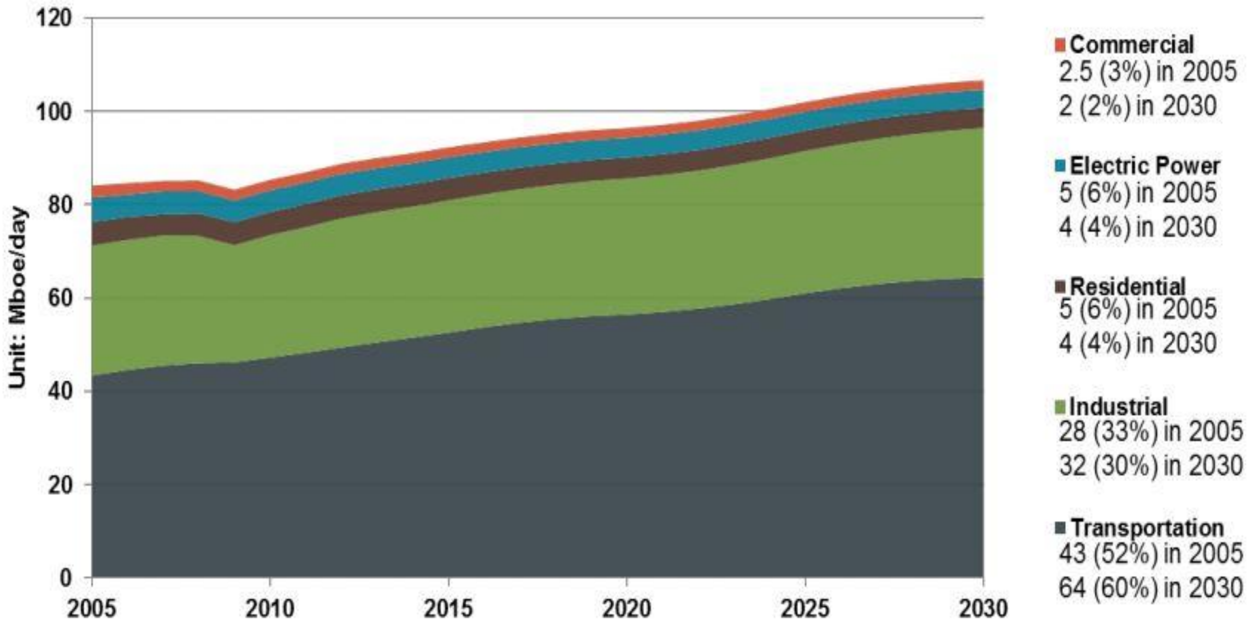


Figure 2.1: Consumption of liquid fuels by sector.

(Source: ICCT, 2014).

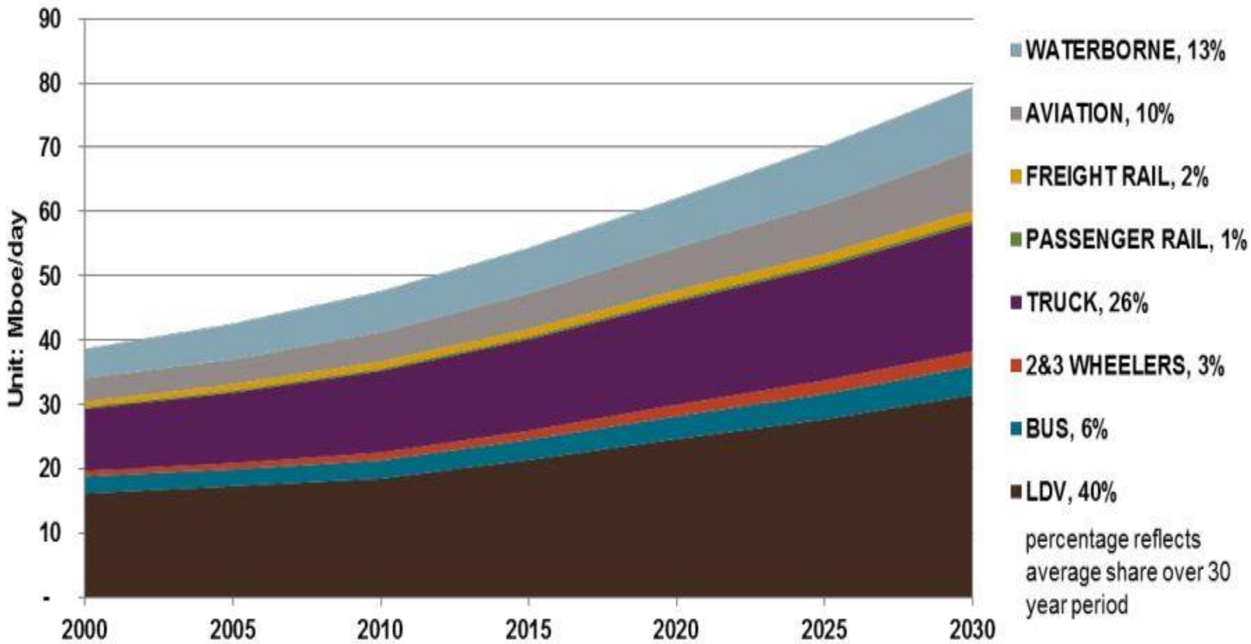


Figure 2.2: Share of energy consumption by transportation mode (vehicle sector) (Source: ICCT, 2014).

2.3 Fuel Efficiency in Automobiles

Vehicle fuel efficiency and emissions are influenced by several factors including vehicle (car body design) design, vehicle size and weight regulation, zone of operation, driver technique and weather factors.

2.3.1 Vehicle Design/Aerodynamics

Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is also a subfield gas dynamics, with much theory shared with fluid dynamics. Aerodynamics is often used synonymously with gas dynamics, with the difference being that gas dynamics applies to all gases. Understanding the motion of air (often called a flow field) around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy can be defined and used to solve for the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis. (Krishnani P.N, 2006)

2.3.2 Aerodynamic Forces acting on Moving Vehicles

(i) Drag Force

Drag Force acts on a body moving through fluid in order to resist its motion through the fluid. As stated in the research work of Janosko et. al (2017), air resistance is the result of normal air pressures on body surface and friction forces acting in the tangential direction of air flow around the bodywork. The undesired pressure above and below the vehicle (high vacuum under the vehicle, slight overpressure under the vehicle) causes the air to swirl in the transverse plane of the vehicle (so-called induced air resistance), which is another component of the overall air resistance

Krishnani P.N (2006) stated that the drag force constitutes about 65% of the total force acting on the complete car body. The total drag force is a function of the density of air, speed of the car, value for the coefficient of drag and the car's frontal area The Aerodynamic drag force can be computed from the formula stated in equation (1).

$$F_D = \frac{1}{2}\rho \, C_D v^2 A \text{ ------------------------ (1)}$$

Where:

F_D = Drag Force

$\boldsymbol{\rho}$ = Air Density

C_D = Drag Coefficient

A = Frontal Area of the Vehicle

V = Velocity of the Vehicle

(ii) Lift Force

Just as the drag force acts in the horizontal direction to slow down the vehicle, a vertical force known as the Lift force tends to lift a vehicle off the ground while in the negative vertical direction, it can also result in down force. Automotive Design requires keeping the value to an optimum value without excessive down force or excessive lift.

$$\boldsymbol{L} = \frac{\mathbf{1}}{\mathbf{2}} \rho V^2 C_L A \text{ ----------------- (2)}$$

Where:

$\boldsymbol{\rho}$ = Air Density

V = Velocity of the vehicle

L = Lift Force

C_L = Lift Coefficient

A = Frontal Area of Vehicle

2.3.3 Designing for Aerodynamic Efficiency

In this area, the considerations for automotive design include: minimizing frontal area to reduce frontal air pressure, lowering vehicle frame and profile to minimize air flow under the car, converging bodywork slowly so as to avoid flow turbulence and reduce drag, covering open wheels, and using a material for outer shell that reduces surface friction. All of these design considerations will ultimately affect the coefficient of drag associated with the shape and improve the fuel economy.

The drag forces acting on the vehicle during operation can be lessened by designing an aerodynamic profile for the outer shell. The most important factors regarding drag reduction considered during the design process are frontal area, frontal pressure, and rear vacuum effects on the vehicle. One method for minimizing the air pressure at the front of vehicle is to decrease the frontal surface area. This reduction in surface area for the air to hit will result in less air molecules compressing. It was imperative that the frontal profile was aerodynamically shaped to prevent large pressure differences on the surface of the shell. (D'Agostino et al, 2013)

Aerodynamic styling of a car is one of the most crucial aspects of car design-a highly complex phenomenon, encompassing the task of an artful integration of advanced engineering and stylish aesthetics. A lot of emphasis is laid on the aerodynamics in car design as an aerodynamically well designed car spends the least power in overcoming the drag exerted by air and hence exhibits higher performance- cruises faster and longer, on less fuel. (Sivaraj and Gokulraj, 2012)

Peterbilt Motors Company, (2009) presents a white paper on Heavy vehicle aerodynamics and fuel efficiency. This paper reviews the aerodynamics drag losses which make the vehicles to utilize large capacity engines.

Buresti et al, (2007) carried out a research on Methods for the drag reduction of bluff bodies and their application to heavy road- vehicles in which they stated that in order to reduce the bluff body drag, boat- tailing has been applied & this reduced the base drag to about 5% to 10% respectively. Further to reduce the drag force Tractor – Trailer gap has been occupied by a device so that both drag reduction and also the trailer can turn easily around the turns without any clashing. Also fairings and flow-deflection devices have been provided in the pressure drag not only of the axle but also of the trailer base. This paper also points out the wheels of the road vehicles are, in general, a source of considerable aerodynamic drag, therefore in order to reduce such drag wheel housings have been provided. It has also been said clearly that optimization.

Mc Callen (2004) in their experiments found out removal of rear view mirror alone will bring down the drag of the vehicle by 4.5%. Any gap in the vehicle body will result in flow separation and flow circulation. This investigation revealed a reduction in drag value until the front leading edge radii value reaches 150mm.

Panu Sainio (2007) from Aalto University conducted a research on aerodynamics possibilities for heavy vehicles, and they came out with a conclusion that boat tail approach is known to be a good solution in terms of aerodynamics.

The classical boat tail may increase the length of the vehicle significantly, but by using trailing edge blowing there might be possibilities to shorten the solution and still to have the full aerodynamically performance of the long boat tail.

Edwin J. Saltzman and Robert R Meyer, (2007) carried out studies on reducing the drag of trucks and buses. The final model equipped with rounded horizontal and vertical corners, smoothed under body and a boat tail achieved a coefficient of drag (Cd) value of 0.242.

The drag coefficient caused by air flow over different shapes is shown in Figure 2.3. Figure 2.4 shows the drag force acting on a conventional car.

2.3.4 Effect of Aerodynamic Drag on Fuel Efficiency

Ahmed (1999) researched into the effect of reducing the drag, mass and power on the fuel consumption of two category of vehicles based on their power. The graphical result is shown in the Fig. 2.5. The result shows that reduction in drag, mass or power has significant effect on fuel consumption. It also reveals that reduction in drag is more efficient than reduced mass or installed power.

The Society of Automotive Engineers in her published book on Aerodynamics (SAE, 1998) revealed that aerodynamic drags contribute up to 11% of vehicle fuel consumption (Fig. 2.6)

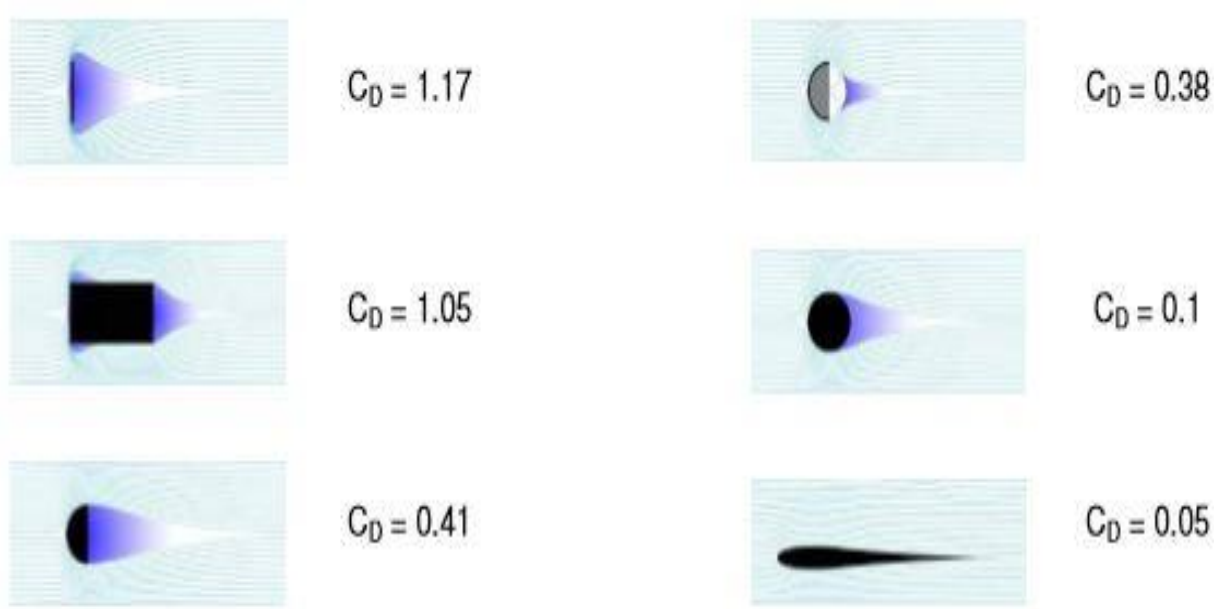


Figure 2.3: Drag Coefficients of different shapes

Figure 2.4: Drag Force acting on a car

Car Category	Mass [kg]	Power [KW]	Fr. Area [m²]	C_D
A	800	40	1.76	0.36
E	1550	135	2.15	0.33

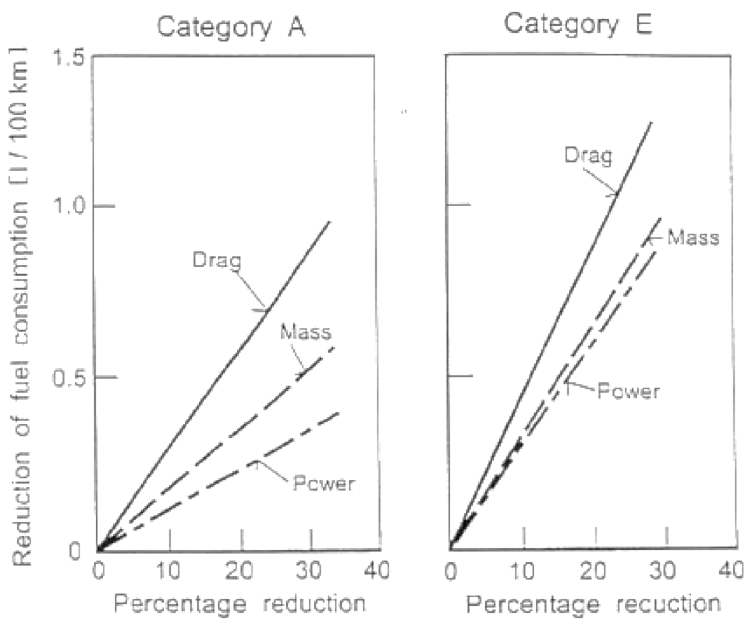


Figure 2.5: Impact of Drag, Mass and Power Reduction on Fuel Consumption

Source: Ahmed (2009)

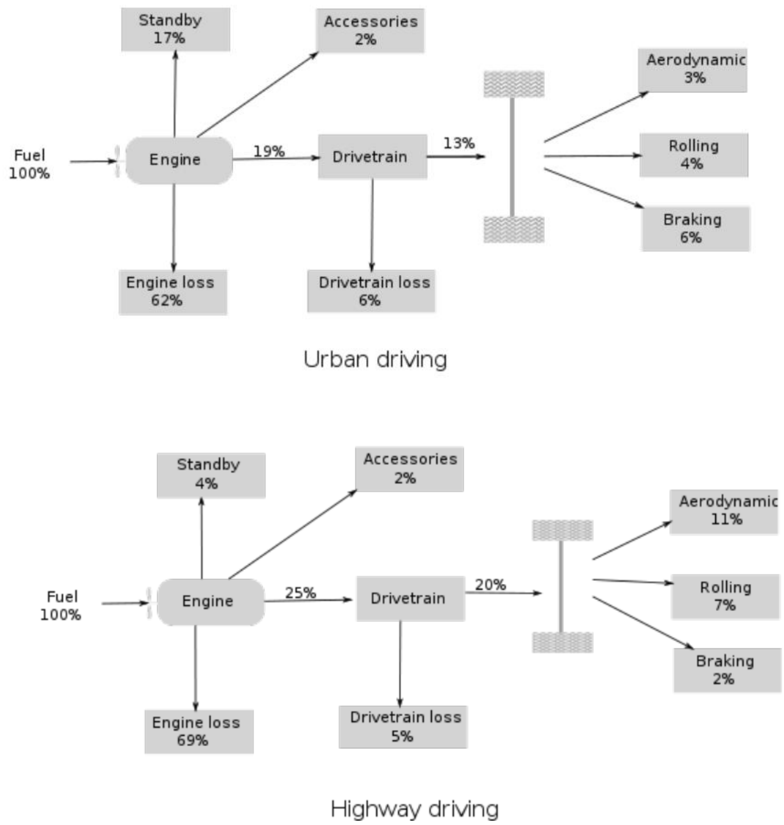


Figure 2.6: Typical Fuel Energy Usage at Urban and Highway Driving

(Source: SAE, 1998)

2.4 Review of Related Studies

2.4.1 Computational Study of Wind Angle on Drag and Lift Coefficient (Zheng, 2009)

In the study carried out by Zheng (2009), the drag and lift coefficients are used in order to compare the different effects of the front window angle on the performance of a car body. The dimension of the car body was 435 x 168 x 148mm (length x width x height) with a 360,000 mesh size. The flow study was carried out with different relative angles between the wind and the car axis ranging from 0˚ to 30˚ as shown in Figure 2-10.

2.4.2 CFD Analysis of Drag and Lift Coefficient on Sedan and Hatchback Vehicle (Salleh, 2009)

The simulation and analysis carried out by Salleh (2009) showed the difference in drag and lift coefficients between two basic vehicle types; a sedan and a hatchback. COSMOS Floworks software was used for the process simulation and analysis of the cars. The result showed that both the drag and lift coefficient for the hatchback design was lower than that of the sedan design. It was therefore observed that the hatchback design was more efficient due to its lower drag coefficient.

2.4.3 Design and Construction of Urban-concept Car Exterior for Shell Eco-marathon 2011 (DaSilva et. al, 2014)

The design and fabrication of an urbanconcept car that participated in Shell Eco-marathon Asia 2011 carried out by M.Bernabe and some other students in 2011 was reported by DaSilva et. al (2014). The design of the exterior body had an average drag coefficient of 0.47.

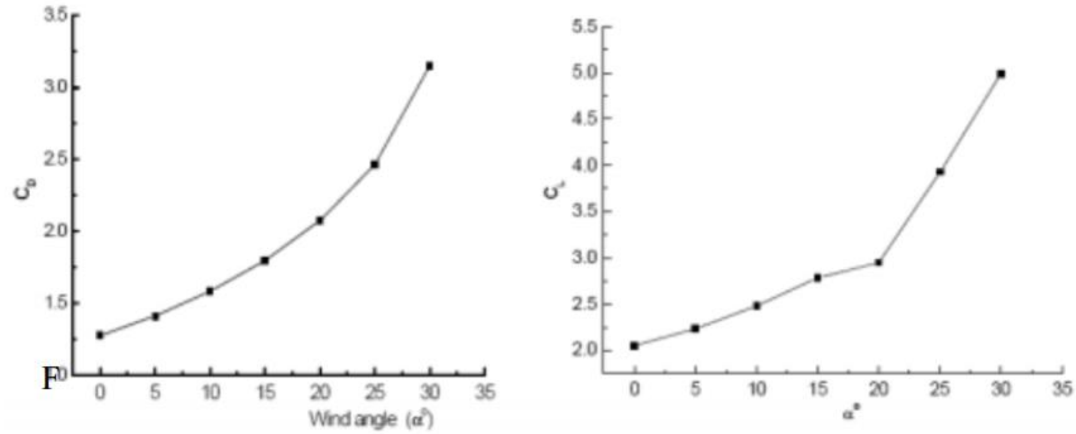


Figure 2.7: Relation between wind angle on lift and drag coefficient

(Source: Zheng, 2009)

2.4.4 Review of CFD Analyses done on Aerodynamic Reduction of Automobile Car (Singh et. Al, 2014)

The researchers did a thorough review of literature being done on the CFD Analysis of Automobiles for drag reduction. It was realized that there are many thrust areas, such as shape optimization, meshing type and boundary condition, that affect the CFD analysis of drag coefficient. It was however revealed that CFD analysis offers superior capability than experimental approach in terms of post processing of data and graphical representation of flow analysis.

2.4.5 Design of Aerodynamic Car body for Shell Eco-Marathon 2015 (Da Silva et.al, 2014)

Da Silva et. Al (2014) carried out a study to design and simulate an aerodynamic car body for the shell eco-marathon 2015 named "Harribon" following the shortcomings of the institute's first urbanconcept entry in 2011 named "Habagat." The researchers used the Autodesk Inventor software for their CAD design and the Autodesk Flow design for their flow simulation. The coefficient of drag was 0.47 for the old design of Habagat; improved to 0.31 after the re-design and simulation of Haribon (shown in Fig. 6).

Figure 2.8: Team Cardicals 2014 Urbanconcept Haribon

CHAPTER THREE

METHODOLOGY

3.1 Method of Approach

Numerous modifications can be made to a vehicle to increase its fuel efficiency. Our approach to designing a vehicle more efficient than the common road vehicles is to focus on changes in aerodynamics as the key factor to provide the greatest increase in efficiency. The aerodynamic efficiency of a car's shape is measured by its coefficient of drag (generally known as its C_D value). For example, a flat plate held at right angles to the airflow has a C_D value of 1.25, whereas the most efficient production car shapes at the moment have a Cd of about 0.28 (how a car works).

However, this C_D value cannot be used by itself to calculate a car's aerodynamic drag because it does not take into account the car's frontal area. The frontal area is the car's total cross-section, or the total amount of space it occupies when viewed from the front. A full-size car and a scale model of the same thing would both have the same C_D value, but the larger version would need a lot more power to propel it at a particular speed because its frontal area is larger. Therefore, the total drag force is a function of the density of air, speed of the car, C_D value and the car’s frontal area. The systematic process flowchart is shown in Figure 3.1.

3.2 Design Requirements

The following sub-sections will outline some specific needs and constraints relating to the design and construction of the vehicle, in accordance to the Shell Eco-Marathon rules and standards.

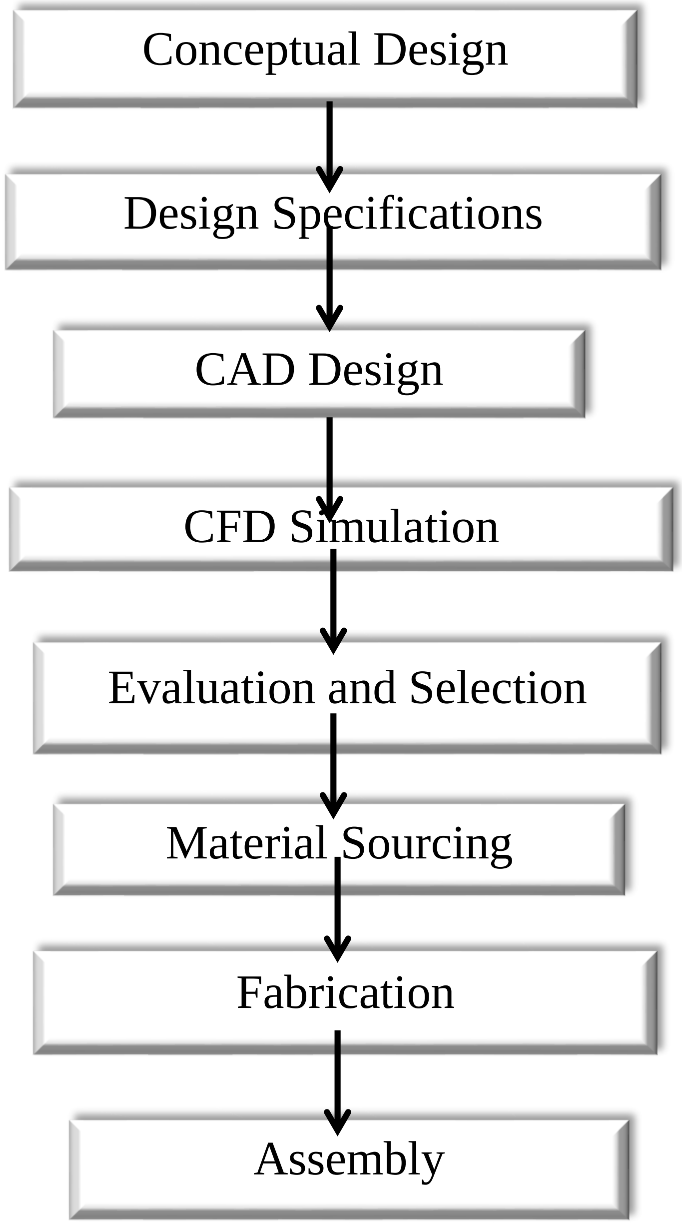


Figure 3.1: Construction Process Flow Chart

3.2.1 Dimensional Constraints

The following dimensions are sourced from the Shell- Ecomarathon Rules for Urban-Concept Vehicles (Appendix A)

- Length: 220cm Minimum; 350cm Maximum
- Width: 120cm Minimum; 130cm Maximum
- Height: 100cm Minimum; 130cm Maximum
- Track Width: 100cm Minimum
- Wheelbase: 120cm Minimum
- Height/Width Ratio: 1.25 Maximum
- Ground Clearance: 10cm Minimum

3.2.2 Design Constraints

- The chassis must incorporate a roll bar that extends 5cm above the drivers head, and past the width of the drivers shoulders with the driver in the standard driving position with the seatbelts fastened. The roll bar must be able to withstand a 700N load without deflecting.
- The vehicle fairing must cover all drivetrain associated parts.
- The cover around the engine must be easily removable to facilitate inspection access
- Vehicle with wheels mounted inside the faring must have a bulkhead that separates the wheels from the driver.
- The vehicle must have a full floor that will prevent the driver from any contact with the ground at any point during normal operation
- Vehicle windows must be made from a material such that in the event of an impact, they do not break into smaller shards.

- The vehicle fairing must not impede driver visibility directly ahead of the vehicle or 90 degrees to either side of the vehicle's longitudinal access.
- Any active aerodynamic apparatus are specifically prohibited.
- Vehicle must be designed to allow the driver to vacate the vehicle in less than 10 seconds, starting from a fully harnessed position.
- The driver access portion of closed body vehicles must be easily accessible from both inside and outside of the vehicle and must be possible to open without tools. Exterior latches must be clearly marked with red arrows.
- The Driver's compartment must have a minimum height of 88 cm and a minimum width of 70 cm at the Driver's shoulders.

3.2.3 Aerodynamic Design Considerations

The drag forces acting on the vehicle during operation can be lessened by designing a more aerodynamic profile for the outer shell. The most important factors regarding drag reduction considered during the design process are frontal area, frontal pressure, and rear vacuum effects on the vehicle. One method for minimizing the air pressure at the front of vehicle is to decrease the frontal surface area. This reduction in surface area for the air to hit will result in less air molecules compressing. It was imperative that the frontal profile was aerodynamically shaped to prevent large pressure differences on the surface of the shell. As a designer, you want to create a streamline that cuts through the air at speed, and prevent turbulent flow. Another consideration was the effect of flow detachment from the rear profile of the vehicle. Flow detachment is the rear vacuum that occurs when vehicles produce holes left in the air when they operate. More aerodynamic shapes will reduce the vacuum area behind the vehicle. The designs we are considering all have the rear profiles converging to one point. Since the car is to be designed as an urban-concept vehicle; which would have speed limits while traveling on the road, lift and down-force effects will be negligible for the final design process. Down-force is necessary for race car design due to the high cornering and straight line speeds, but our vehicle is not built for such speed. Lift effects are also not necessary either, because the speeds are not high enough to produce sufficient lift forces to overcome normal forces. Goals to be focused on achieving during the design stage are as follows:

- Minimizing frontal area to reduce frontal air pressure,
- Lowering vehicle frame and profile to minimize air flow under the car,
- Converging bodywork slowly so to avoid flow turbulence and
- Reduce drag, covering open wheels, and using a material for outer shell that reduces surface friction.

All of these considerations will ultimately affect the coefficient of drag associated with the shape and improve the fuel economy.

3.2.4 Concept Design

For the car concept, the computer design was done using the software: Autodesk Inventor.

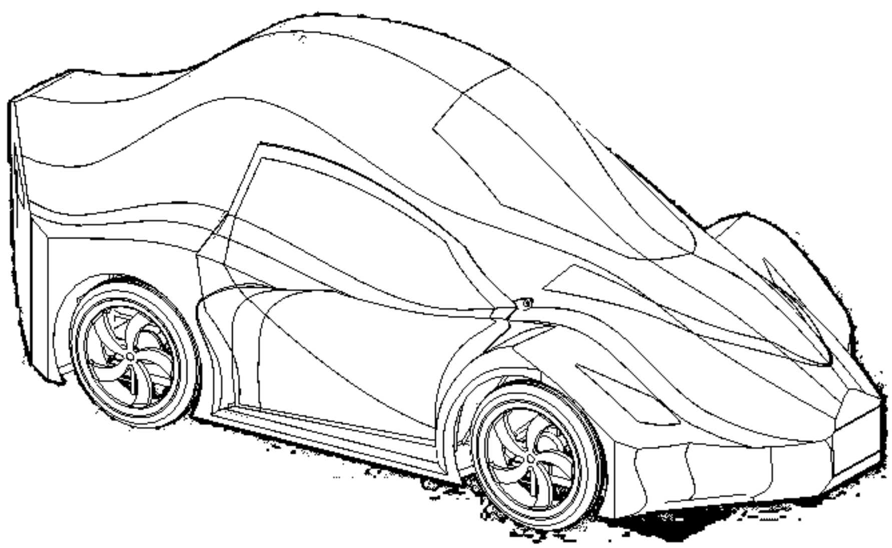


Figure 3.2: CAD Design of the Initial Concept

3.3 Test/Simulation

After the design of various models, they were subjected to Computational Fluid Dynamics Simulation using the ANSYS Fluent 16.2 Software and Autodesk Flow Design 2016 in order to test for the aerodynamic nature of each one of them.

3.3.1 Aerodynamic Test

One method for obtaining the coefficient of drag of various vehicle designs is utilizing a wind tunnel which requires physical models of the designs considered, scaled down to about 1:15 size.

Car manufacturers use wind tunnels to examine how their car prototypes would work. Usually in a wind tunnel, the car is anchored down and a stream of air is blown past it to simulate the conditions that the car would meet when driven forward. The flow of air past the car is made visible by attaching small tufts of wool to the car's body, or by blowing a stream of smoke past it. (how a car works).

In both cases, the path that the wind takes as it flows over the car can be seen by how the wool or smoke behaves. Smoke also shows the behaviour of the air in front of and behind the car. Woollen tufts arrange themselves along the lines of the airflow over the body but cannot show the air behaviour in front of or behind the car. The model or car in the wind tunnel can be turned round at various angles to the airflow so that the engineers can see how the body shape behaves in side winds. However, due to the inaccessibility of a standard and suitable wind tunnel for this project, an improvisation will be made using a virtual CFD analysis.

3.3.2 Selecting the most efficient design

Six proposed concept designs were designed; which was simulated to test for their aerodynamics after which the selection of the final concept decision was chosen using a matrix.

Criteria for selection include:

- Aerodynamic efficiency (less drag)
- Ease of fabrication
- Material Cost/Availability
- Strength to weight ratio

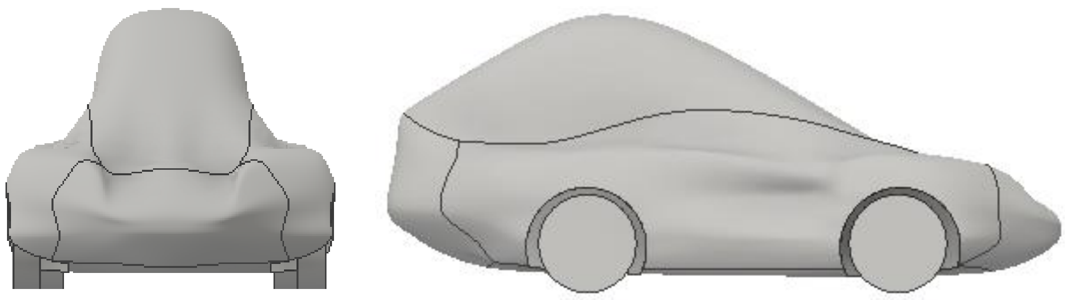
3.3.3 Aerodynamic Analysis and Simulation

Having designed the shell models using Autodesk CAD software, a virtual flow simulation was then performed on Computational Fluid Dynamics (CFD) analyses using ANSYS Fluent 16.2 or Autodesk Flow Design.

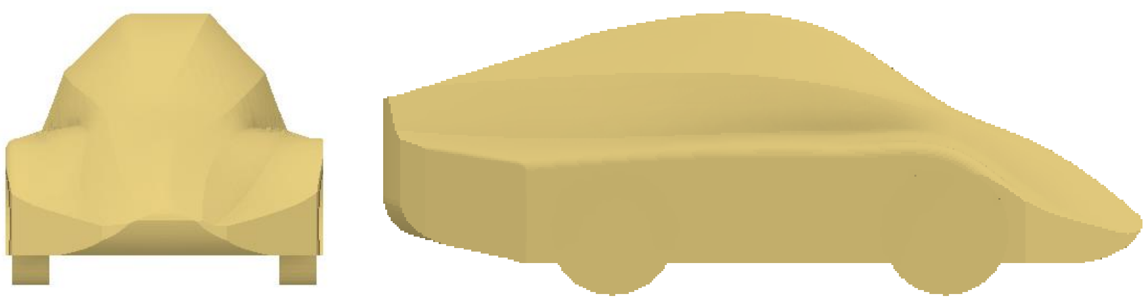
The method for correctly sizing the dimensions of the shell involves importing the CAD frame design and forming a shape around it; and then running it through the CFD. The CFD Software will serve as the primary method to obtain drag forces and drag coefficients of the considered designs.

A typical flow analysis, using air as the active fluid, over a car modeled and imported into CFD program, is as shown in the Figure 3.4.

Figure 3.3: CAD Design of Concept Models



Model 5



Model 6

Figure 3.4: CAD Design of Concept Models (cont’d)

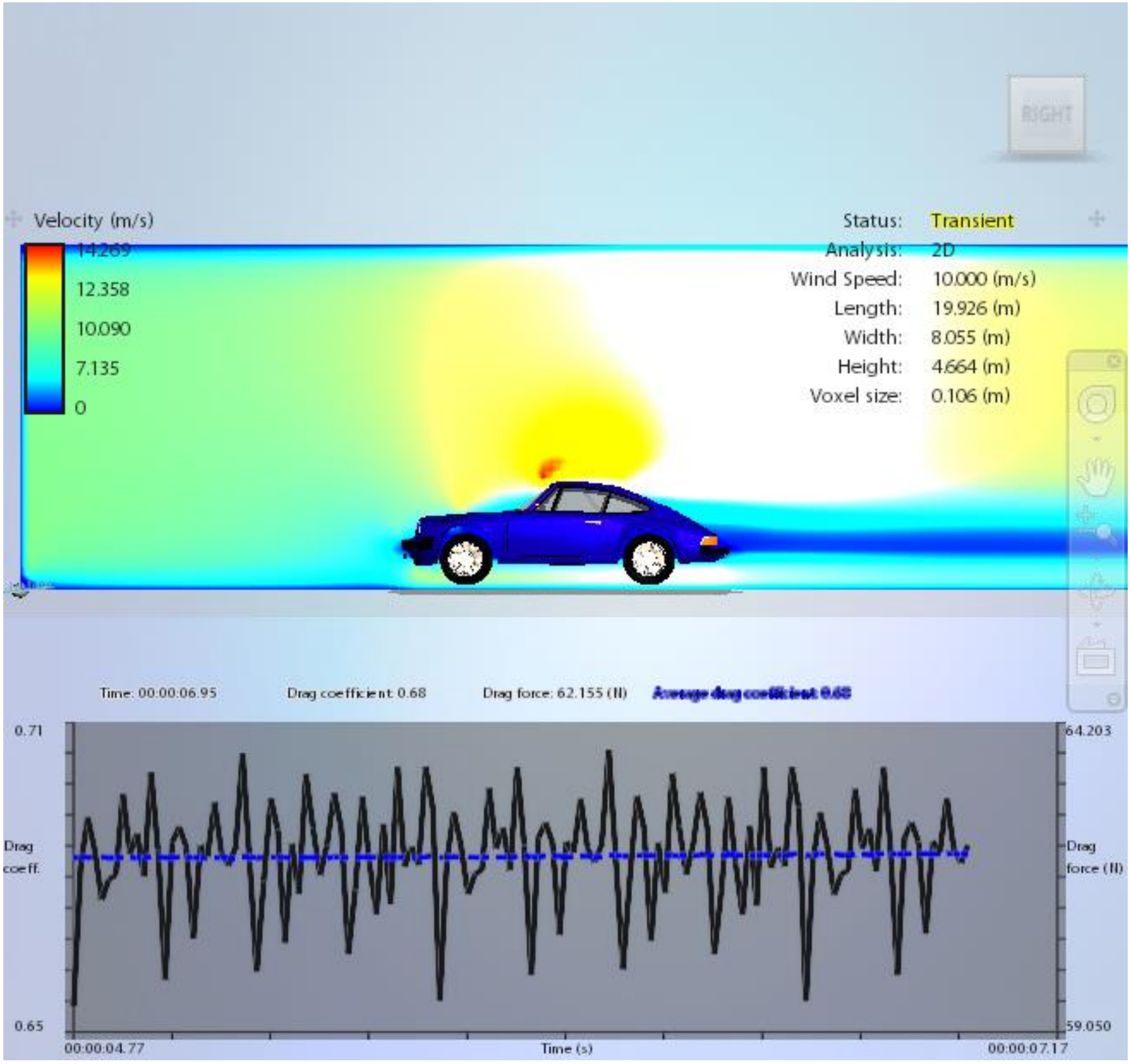


Figure 3.5: CFD velocity profile and drag coefficient for a conventional car (velocity of air = 10m/s)

3.3.4 Determining the Aerodynamic Efficiency

Aerodynamic efficiency of various cars can be compared with the formula that gives the coefficient of drag acting on a body:

$$Cd \; x \; A \qquad \text{Eqn 3.2}$$

Since this coefficient is based on two factors:

- The drag coefficient
- The frontal area

To design a vehicle of less drag coefficient, the model has to be designed to be aerodynamically ‘slippery’.

To arrive at the coefficient of drag, the design of the models developed will be simulated on CFD (Computational Fluid Dynamics) using Autodesk Inflow and Ansys softwares.

In addition, the surface area of the car needs to be reduced as much as possible. To achieve this, we aim at reducing unnecessary space requirements in the vehicle. The vehicle is also designed to be single-seater. The frontal area of each model will be calculated from the Autodesk design information.

3.4 Aerodynamics Simulation in CFD

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that utilizes the use of computer to simulate the flow of fluids (liquids or gases) around objects defined by boundary conditions. CFD softwares uses numerical methods and algorithms to process such simulation. The CFD software utilized in this project is ANSYS Fluent and Autodesk Flow Design.

The softwares have three basic elements that divide the complete analysis of the numerical experiment to be performed on the specific domain or geometry; which include:

(i) Pre-processor

(ii) Solver and

(iii) Post-processor

These elements therefore differentiated the steps of carrying out the CFD simulation into three phases; which are further highlighted below.

3.4.1 Pre-Processing

The pre-processing requires the use of computer application software for the input of the flow problem in a format understandable by the solver. Some activities required of the user at this stage include:

i. Geometry Creation: The objects that participate in the CFD analysis are to be modeled using Computer Aided Design (CAD) software. Out of the various ones available such as Solidworks, Pro-E, CATIA, etc. Autodesk Inventor Professional 2016 had been chosen to be used for this thesis. The design and optimization processes revolved round the use of this CAD software. The three-dimensional modeled vehicle is saved in a STEP file format for import into the CFD software.

ii. Meshing: During meshing, the defined region around the geometry is divided into several structured elements. The elements are connected to each other through nodes; grids are thereby generated. CFD software uses this to aid its finite volume numerical simulation. Some softwares specialize in performing only an aspect of the meshing. For example, 'GAMBIT' creates surface meshing, 'T-grid' creates volume meshing while ANSYS does all encompassed.

iii. Definition of Parameters: The fluid properties are to be inputted at this stage. The fluid considered is air, the flow is further considered incompressible, which means the density is taken as being constant. The speed of the air is taken as 10m/s, the mean speed for which the vehicle was designed.

iv. Boundary Conditions: The boundary conditions for the mesh are assigned finite values at this stage. These boundary conditions initializes the CFD domain and could be different for different setups.

3.4.2 Numerical Solver

Numerical solving constitutes the major aspect of CFD processes. Finite difference method, Finite element method and the finite volume method are diverse methods used by different CFD solvers. The finite volume method is however used in our CFD analysis as the finite difference and element method are only suitable for stress and structural analysis and not appropriate for aerodynamics flow.

The finite volume solving is carried out in the computer by the CFD software firstly integrating the governing equations of fluid flow over all the control volume within the meshed region. After which, it converts the integral forms of the equations into a system of algebraic equations. It then performs the calculation of the algebraic sum through iteration.

3.4.3 Post-processing

The post-processing uses the versatile data visualization tools of the CFD solver to display the results of the simulation in form of vector plots, shaded contour plots, graphs or other display formats.

3.5 Material Selection

Several materials can be found in the market for the car build. However, some important properties have to be considered such as the strength-to-weight properties, availability, purchase cost, machinability, and a reasonable application/ fabrication time. The Table 3.1 shows the initial shell material considerations

Monokote is a plastic shrink wrap that is used to cover remote controlled airplanes. This material is extremely lightweight and fairly puncture resistant however, it requires an internal structure to support it. With the curvature of our final shell, fabricating a structure to support the material would be difficult therefore this option would not be chosen. Vacuum forming a plastic such as PETG could dramatically shorten shell fabrication time, but the need of a high temperature susceptible mold, rules out vacuum forming because of costs. In order for the team to create the outer shell using Carbon Fiber, Dacron, or Fiberglass, a mold for our desired exterior design would be necessary. The mold can be used to form that shape of the car when a composite fabric is applied over it. Carbon Fiber offered unnecessary amounts strength at a high price. Dacron was calculated to be more expensive than fiberglass by #2363/cm.

The fabric chosen for the shell material is 4 oz. fiberglass. The choice was a function of the material having a necessary strength to weight ratio as well as cost.

Table 3.1: Summary of potential shell materials

Source: D'Agostiino, et.al. (Calvin College Engineering)

Material	**MonoKote**	**PETG**	**Carbon Fiber**	**Dacron**	**4 oz. Fibreglass**
Layers Needed	1	1	2	2 + Foam	2
Cost	Less Expensive	Least Expensive	Expensive	Very Expensive	Less Expensive
Pros	-Extremely -Lightweight -Easy Application	-Lightweight -Vacuum Formable	-Aesthetic Appeal -High Strength & Lightweight	-Less Technicalities	-Lesser Technicalities -Cost Effective -Easily accessible
Cons	-Low Strength -Need Frame	-Need Stronger Mold -Not easily accessible	-Difficult Application	-Difficult Application	---

3.5 Fabrication process

After much deliberation on the pros and cons of the material selection, the fabrication of the physical vehicle had to commence. With the assistance of some fine art students, we were able to carry out the fabrication in stages, which is highlighted below:

STAGE 1: Sourcing for Clay for the execution of the model.

There are different types of clay, and deciding the actual type for use required making some compromise by comparing their advantages and disadvantages with respect to the fabrication of the model.

- Ball Clay: This is majorly preferred for use when developing a model that needs to be made into any kind of shape because of its flexibility. It is suitable for modelling, but not suitable for work that requires firing. It can be obtained beside streams and rivers.
- Red Clay (Terracotta): This is also plastic but also has the tendency to crack easily especially when it is sprayed from time to time. It can be obtained at any location.
- Kaolin: This is not plastic. It has to be combined with other type of clay to become plastic. It is only suitable for works that will require firing. It can be obtained from well, underground where sedimentation occurs.

Ball clay was chosen because of its plasticity, workability and malleability, i.e. it can be easily made into desired shape.

STAGE 2: Actual Execution of the Model.

- Production of the Pads: During the execution of the clay model, pieces of wood, pieces of metal was used to reduce the quantity of clay that would be used for the work.

Usually, a framework is always used but was not used because the model is balanced on its own, so we had to pad it instead. Pieces of clay is being added until the desired form and shape is achieved, and from time to time, the work was beaten to ensure body contraction and compactness.

- Detailing of the Model: After the achievement of the overall form into its basic shape, we started detailing the form by cutting out the intrinsic parts of the model i.e. the windscreen, tyre space, side mirrors, doors, front and backlight, interior extrusion for plate numbers.
- Smoothening of the Model: After achieving the necessary details, the smoothening process commenced. The work was sprayed with water from time to time and covered with nylon to avoid dryness of clay before completion.

STAGE 3: Mould-making for the Model.

After completion of the model, the wwork was divided into segments with pieces of metal plate. Afterwards, the work was lubricated for easy removal of the mould after drying. Then, a solution of gypsum plaster (plaster of Paris) and water was applied on each of the segments. Then, the solution was left to dry. Given all the mould in form of P.O.P (Plaster of Paris), after which the mould was separated following the segment.

STAGE 4: Casting

In casting, the major material used in this stage is Resin, Accelerator and Catalyst. Having separated the mould, the mould was cleaned and lubricated again for easy removal of the cast after completion. Then, a solution of the resin, catalyst and accelerator was applied on the surface of the lubricated mould. A layer of fibre mat was also applied on the surface of the solution. The solution was left to dry a bit before the second layer of the same solution was applied. This process was repeated until the desired thickness was achieved, after which the work was left to set. After setting, the work is removed from the mould and then coupled together using the same solution with which it was produced. At this stage, it becomes a fiberglass material.

STAGE 5: Finishing and Partination.

As soon as the assemblage of the segments, excesses in the work is sand-papered, the work is then prepared for partination. After finiahing, the work was partinated with white auto-based paint.

STAGE 6: Assembly.

The modeling and casting of the tyres went through the same process as that of the body of the car. Four pieces of the tyre was casted out of the model and then finally fixed to the car after completion. The glass of the car was produced using transparent rubber plate.

CHAPTER FOUR

RESULTS AND DISCUSSION

4.1 Computational Fluid Dynamics

After successful design of the different proposed 3D model of the fuel-efficient vehicle with Autodesk Inventor, Computational Fluid Dynamics simulations were carried out on the vehicle based on the procedure outlined in the preceding chapter. Simulations were carried out using ANSYS Fluent 16.2 and Autodesk Flow Design. The steps in solving the CFD problem is highlighted below:

- Create the model geometry and mesh
- Set up the solver and physical models
- Compute and monitor the solution
- Examine and save the results

4.2 Aerodynamic Simulation

The different models in the methodology were simulated in order to determine the most fuel efficient as a result of the aerodynamic nature of such vehicle, and how well it can withstand air resistance without increasing the consumption of fuel.

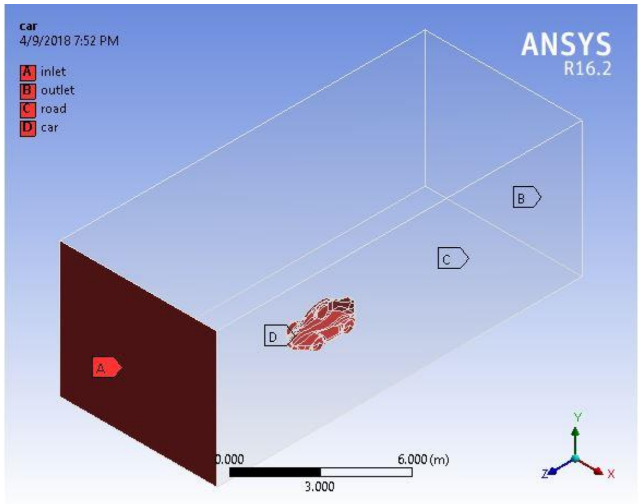
The models were saved in a STEP file format (.stp) in order for it to be read by ANSYS 16.2 simulation software. In the analysis systems of ANSYS 16.2, the Fluid Flow (Fluent) analysis was selected to carry out the simulation. On importing the external geometry file into the design modeler of the analysis, a control volume, somewhat like an enclosure of dimensions 17.205m x 7.2768m x 5.9563m was created around the different models as shown in Figure 4.1–4.6. Subsequently, a Boolean feature was also created in order to subtract the tool body from the target body.

4.2.1 Mesh Generation

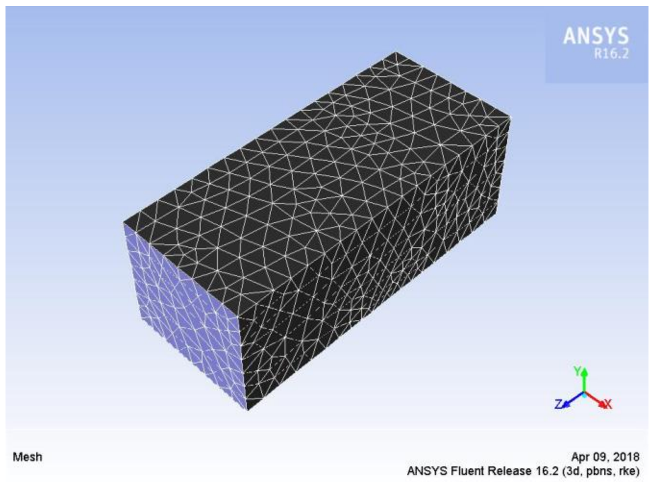
The different models were assigned a minimum element size of 10mm. Refinement and face sizing were assigned in order to achieve an average skewness of less than 0.25. A fine mesh was obtained for each of the predesigned models. The mesh that was generated for each of the designed models are shown in Figure 4.1 – 4.6.

Table 4.1: Mesh Generation Parameters

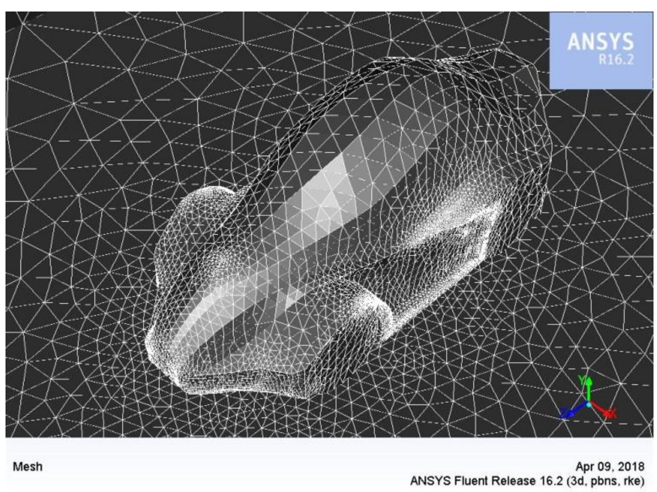
MESH PARAMETERS	Model 1	Model 2	Model 3	Model 4	Model 5	Model 6
Minimum Element Size	1cm	1cm	1cm	1cm	1cm	1cm
Nodes	78243	88218	62104	66514	71367	51378
Elements	431558	486413	338878	362894	389270	282599
Average Skewness	0.23	0.23	0.23	0.23	0.23	0.23
Average Aspect Ratio	1.846	1.8445	1.847	1.8468	1.8505	1.8463
Faces	851243	959341	667009	714230	766072	557184
Cells	431558	486413	338878	362894	389270	282599



(a)



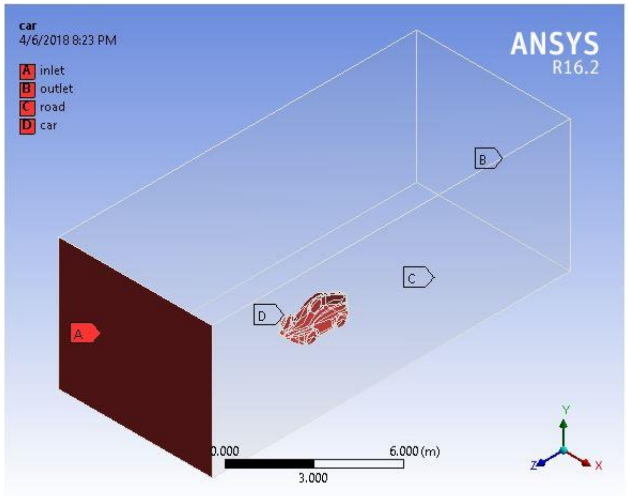
(b)



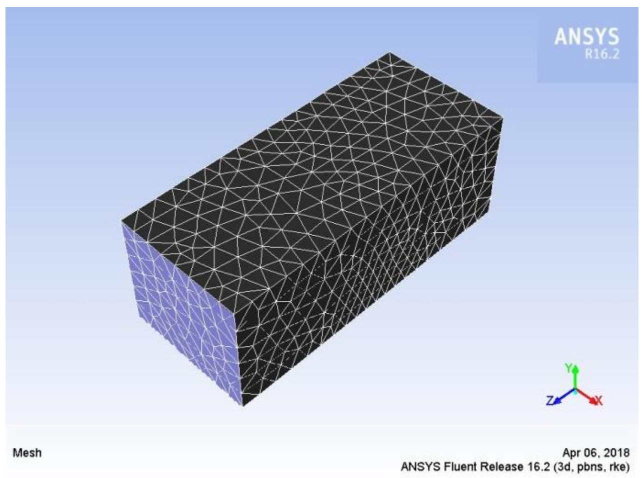
(c)

Figure 4.1: (a) Model1 (b) Model1 Mesh (c) Model1 car Mesh

(a)



(b)



(c)

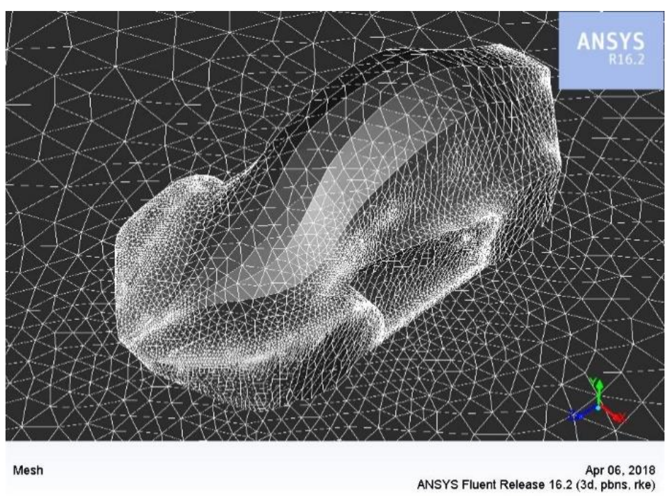
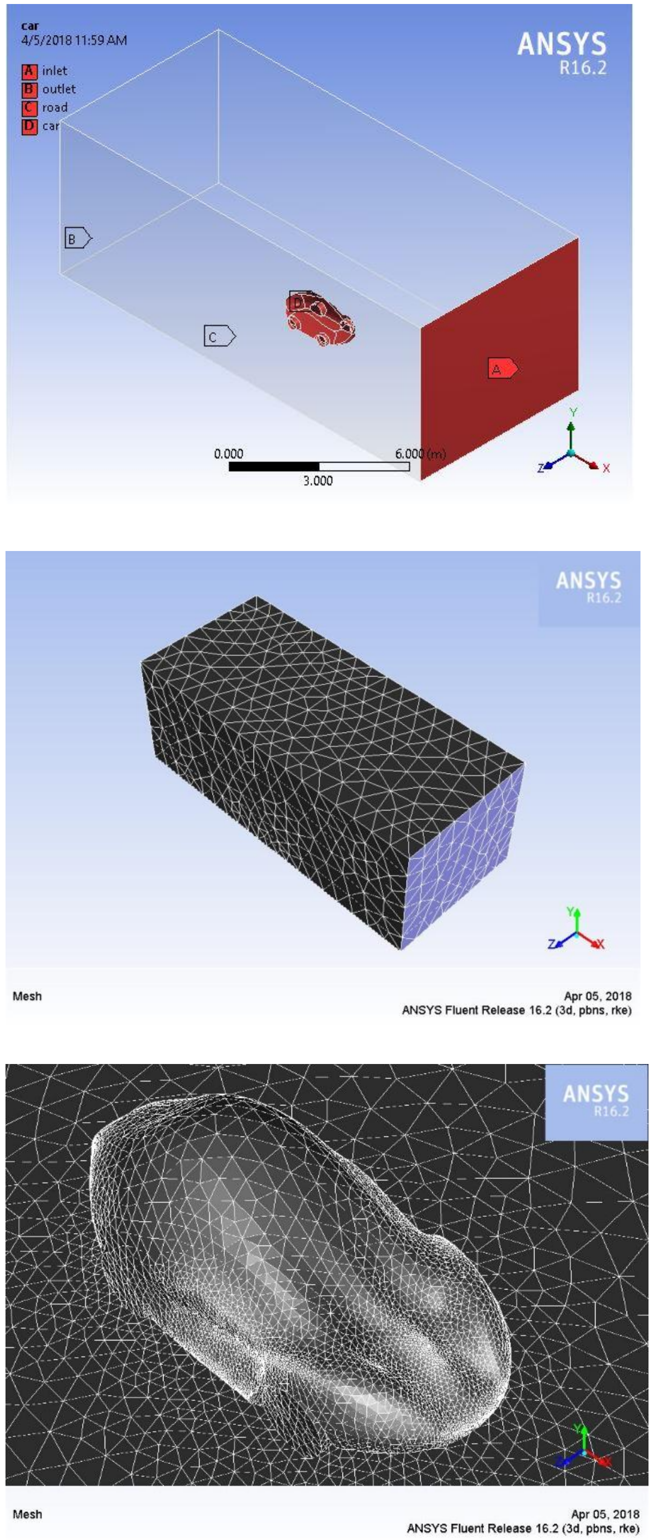


Figure 4.2: (a) Model2 (b) Model2 Mesh (c) Model2 car Mesh

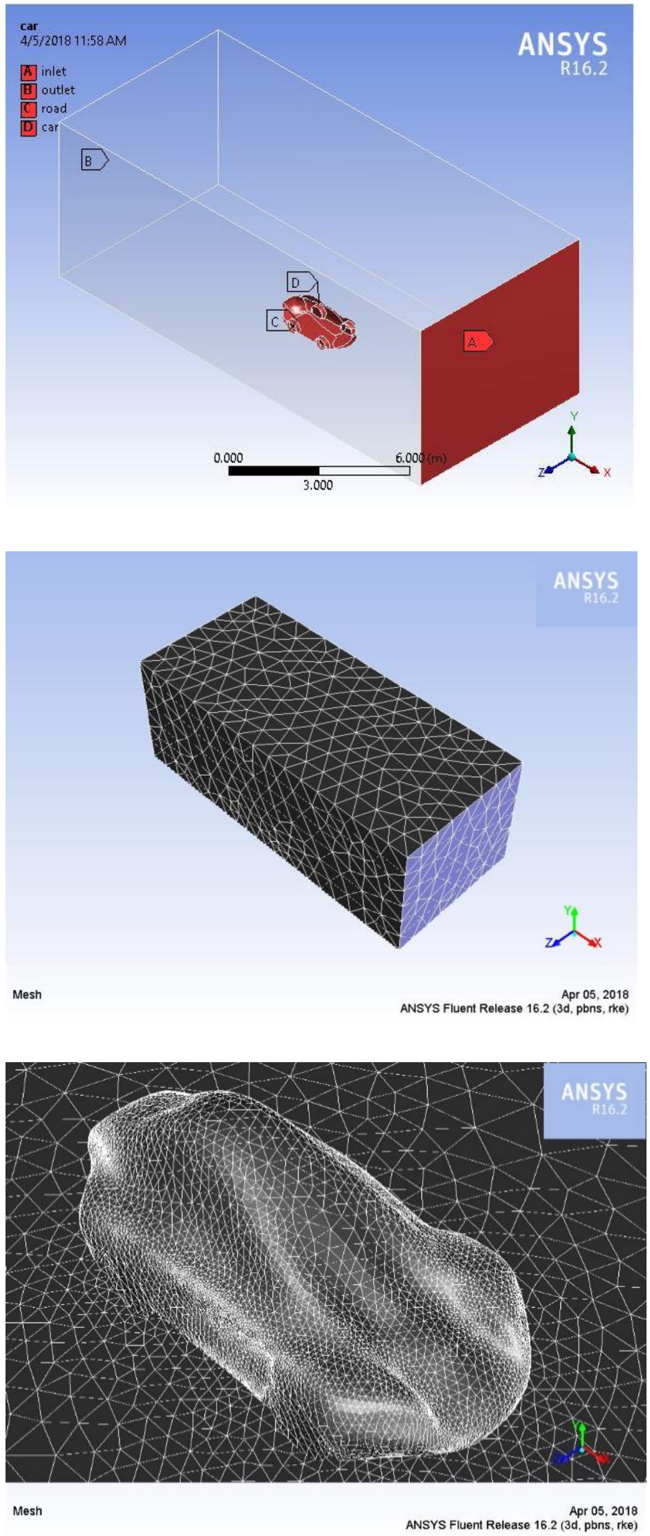


(a)

(b)

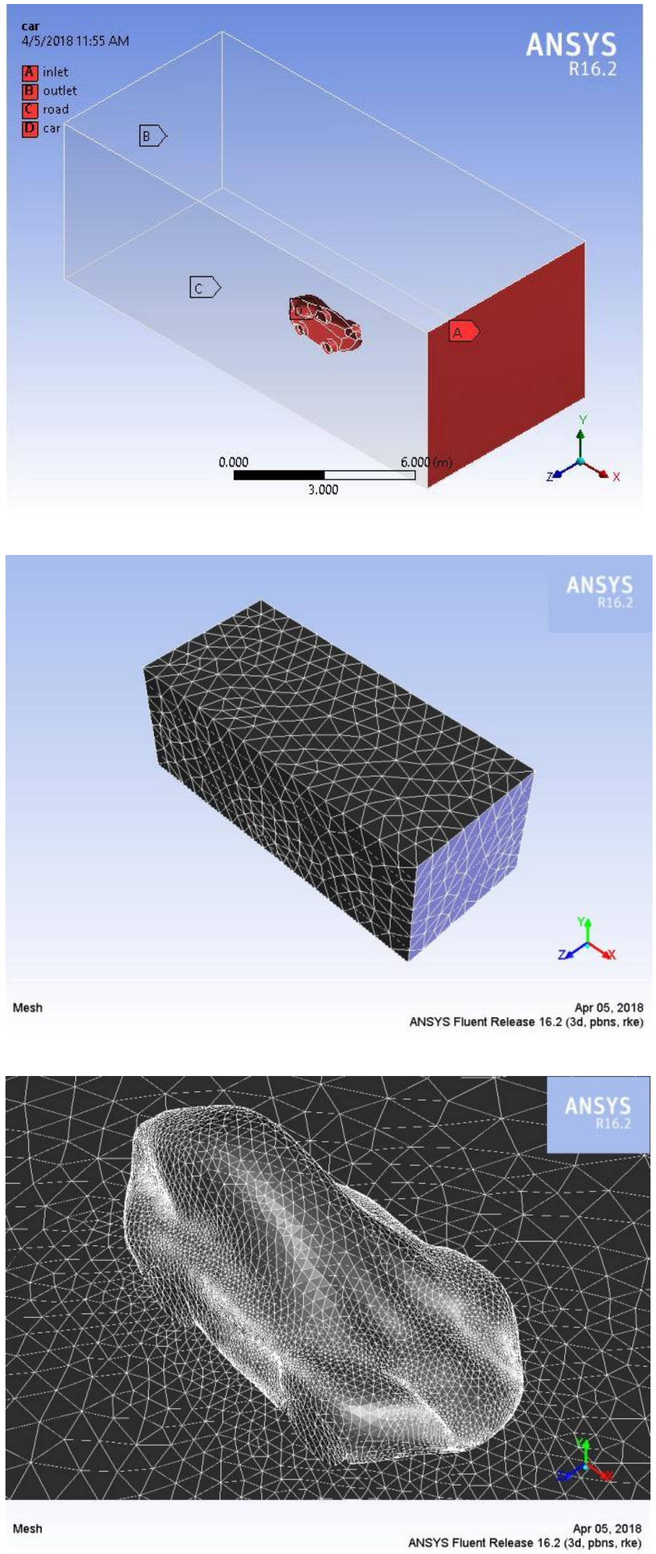
(c)

Figure 4.3: (a) Model3 (b) Model3 Mesh (c) Model3 car Mesh



(a) (b) (c)

Figure 4.4: (a) Model4 (b) Model4 Mesh (c) Model4 car Mesh

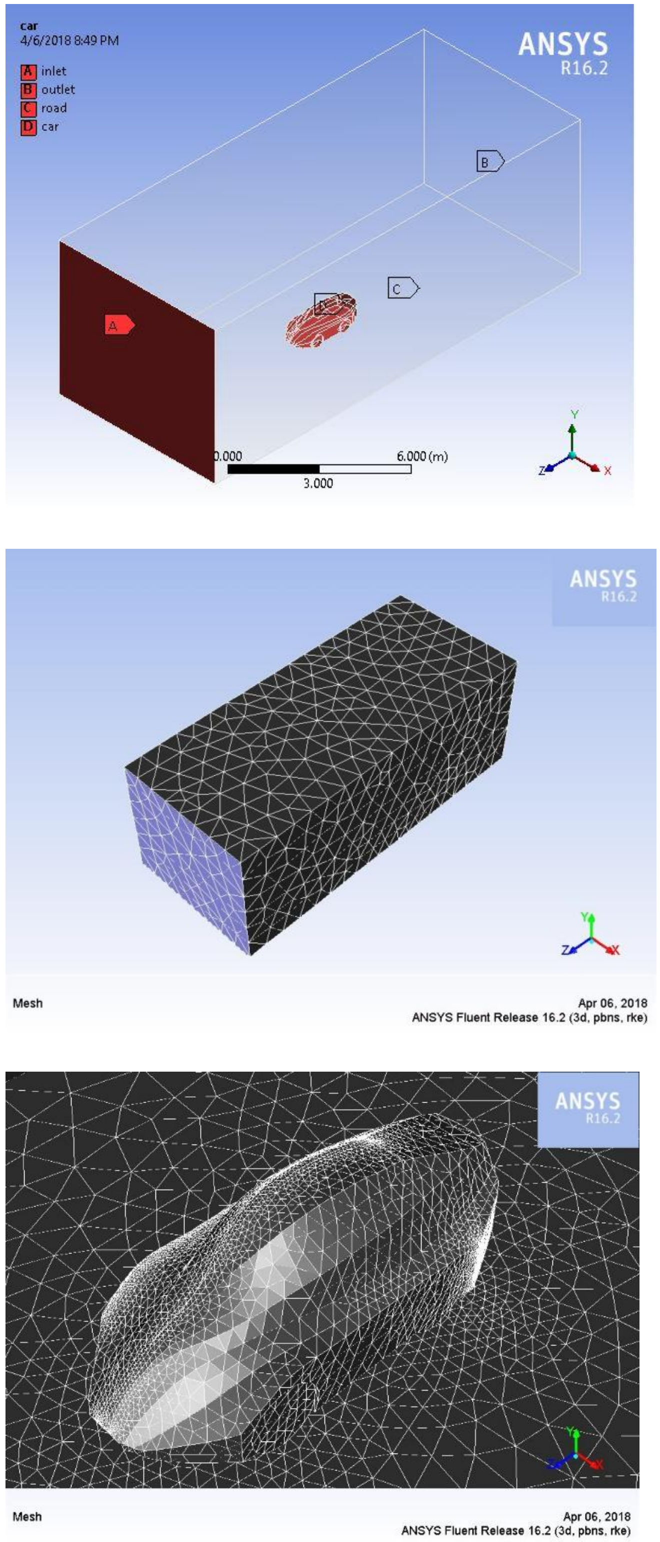


(a)

(b)

(c)

Figure 4.5: (a) Model5 (b) Model5 Mesh (c) Model5 car Mesh



(a) **(b)** (c)

Figure 4.6: (a) Model6 (b) Model6 Mesh (c) Model6 car Mesh

4.2.2 Assumptions

- Air is still; thus the vehicle's speed is taken as the wind speed
- No crosswind acting (thus, no side force or yawing moment produced)
- Standard atmospheric pressure is taken as the pressure at which the vehicle moves

4.2.3 Fluent Setup

After successful mesh generation, the different car models were imported into the ANSYS Fluent Setup, in order to determine their corresponding drag coefficient, lift coefficient and pressure coefficient, and consequently make conclusions on their aerodynamic qualities.

4.2.3.1 'k-ε' model (k-turbulent kinetic energy, ε- turbulence dissipation)

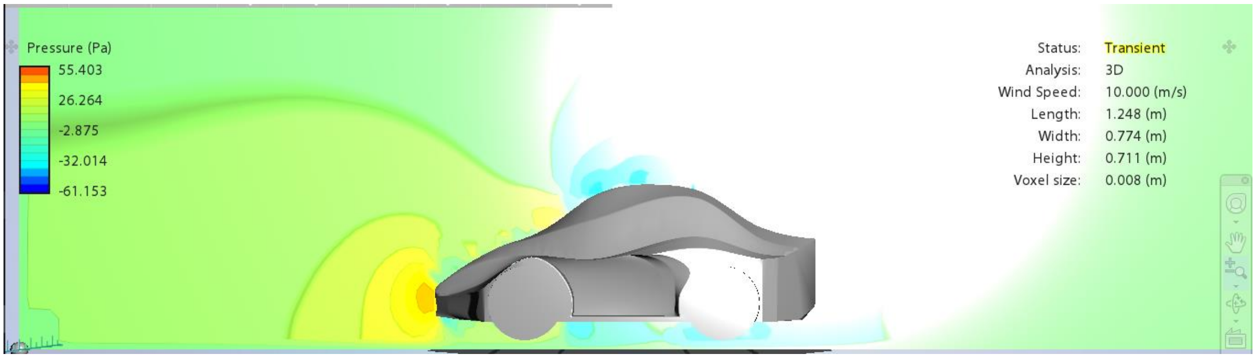
Unlike racing cars with an average speed of 60 m/s, our concentration is not fuel consumption reduction of high speeds vehicles, but rather vehicles running at an average speed of 10 m/s. However, the resulting flow despite this low vehicle speed is turbulent, and as such, a realizable 'k-ε' model has been selected for this analysis.

The k-ε (k-epsilon) model is the industry standard two-equation turbulence model. k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L^2T^{-2}), e.g., m^2/s^2. ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of k per unit time (L^2T^{-3}), e.g., m^2/s^3. The k-e model, like the zero equation model, is based on the eddy viscosity concept. The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. (Petkar et al, 2014)

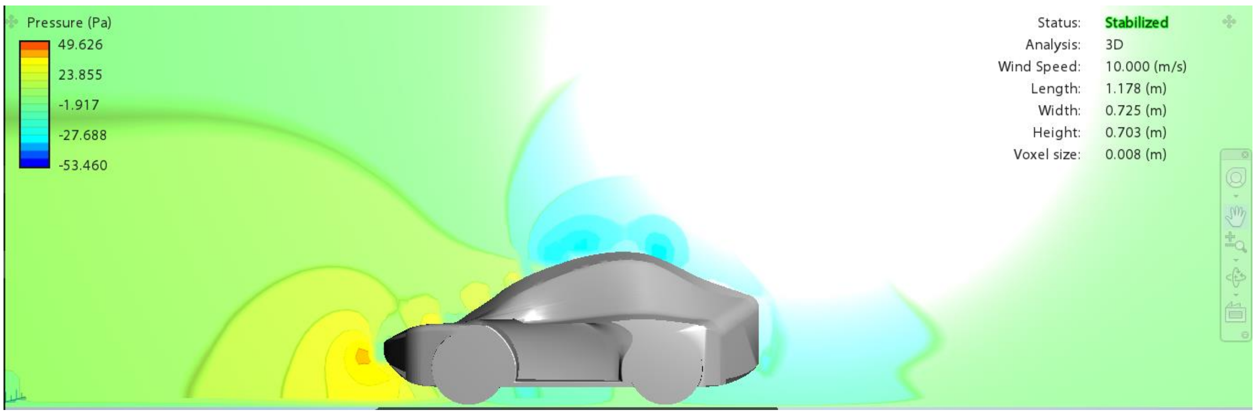
4.2.3.2 Boundary Conditions

Having specified the material and their corresponding material properties, then the named selections in the mesh analysis are given boundary conditions. The inlet of the domain i.e. the air speed at the entrance, “Inlet velocity” = 10 m/s. Car model is “wall” named “car” (in the field “Zone Name”), and it is set as stationary. The sides and top of the enclosure named “wall” is specified stationary wall. Road (in the field “Zone Name”) is specified as moving wall. Since the car does not move, the road will have a velocity in the direction of the airflow, so that the flow under the car can be modeled correctly. Velocity is 10 m/s in the speed field of the road. The outlet of the domain, outlet pressure is set as atmospheric pressure. Therefore, gauge pressure equals 0 Pa, which means the pressure at the outlet equals atmospheric pressure.

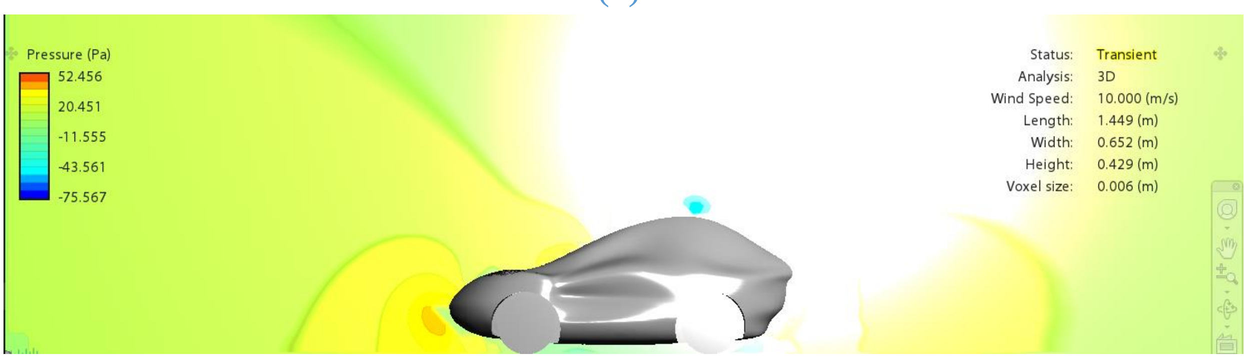
`



(a) Model 1

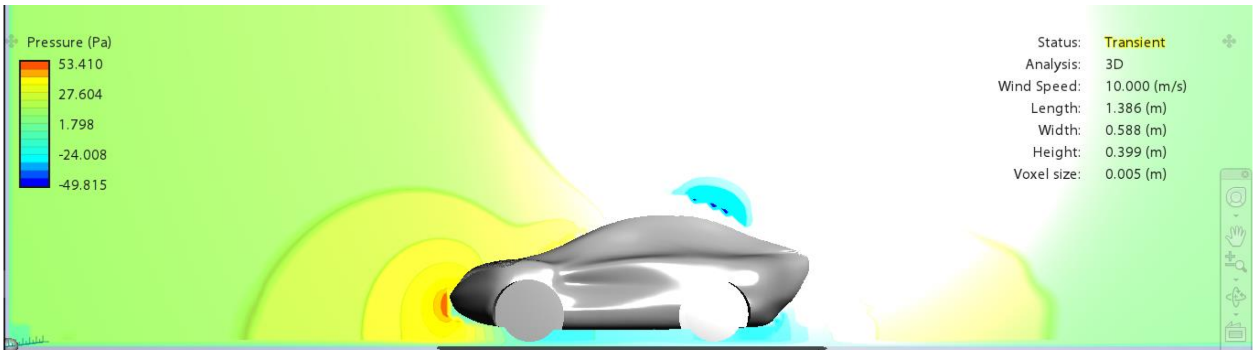


(b) Model 2



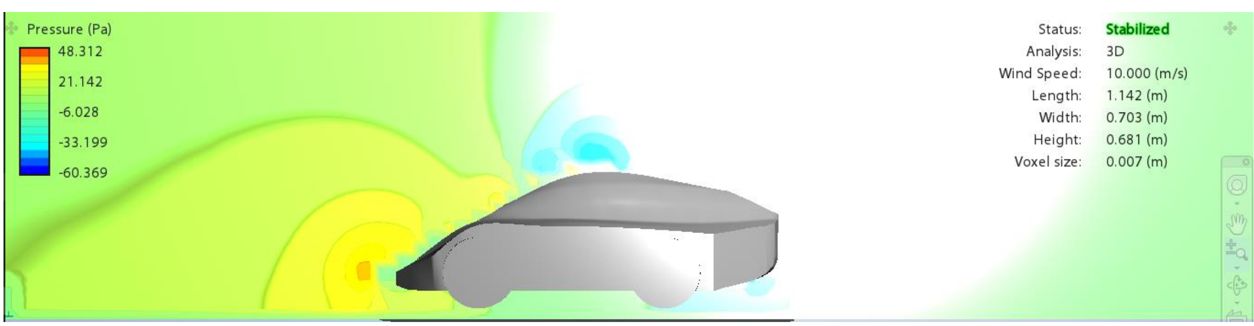
(c) Model 3

Figure 4.7: Pressure Contour Distribution using Autodesk Flow Design



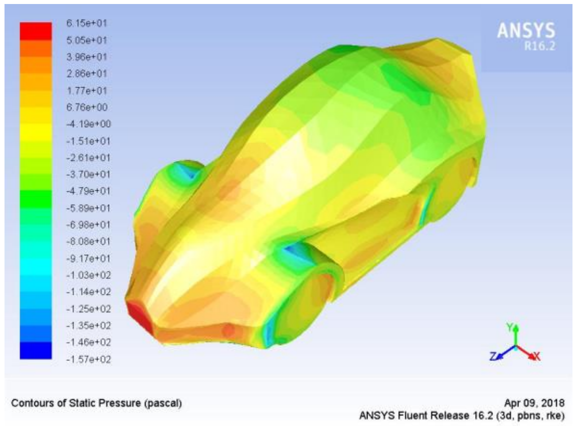
(d) Model 4

(e) Model 5

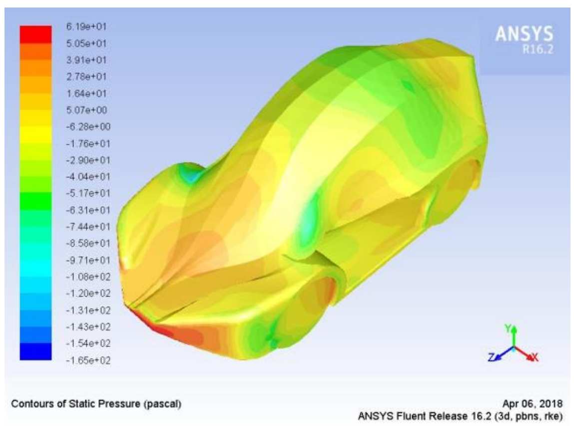


(f) Model 6

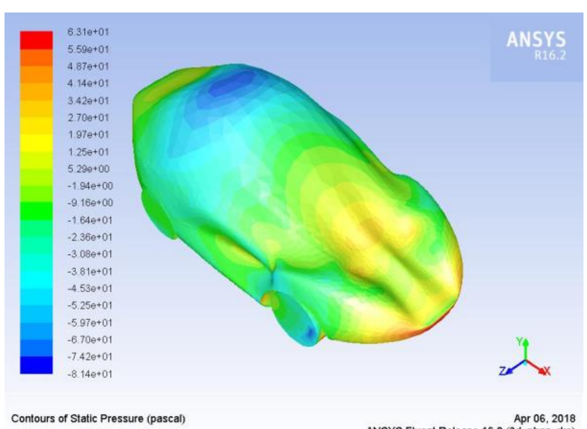
Figure 4.8: Pressure Contour Distribution using Autodesk Flow Design (cont’d)



(a) Model 1

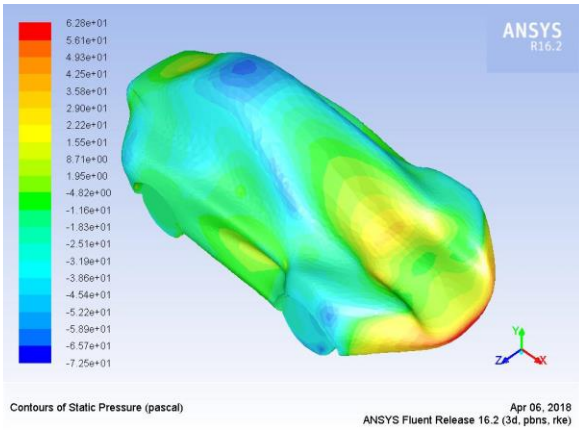


(b) Model 2

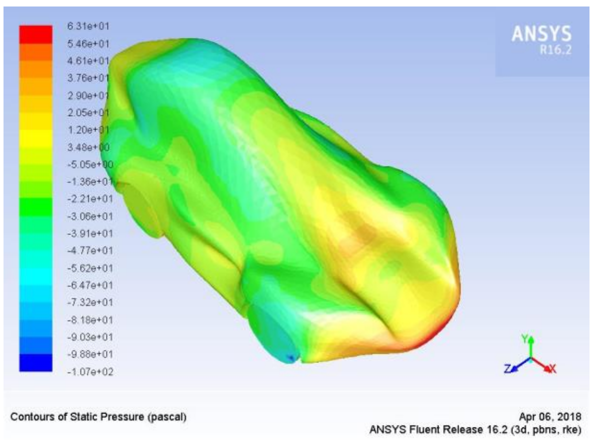


(c) Model 3

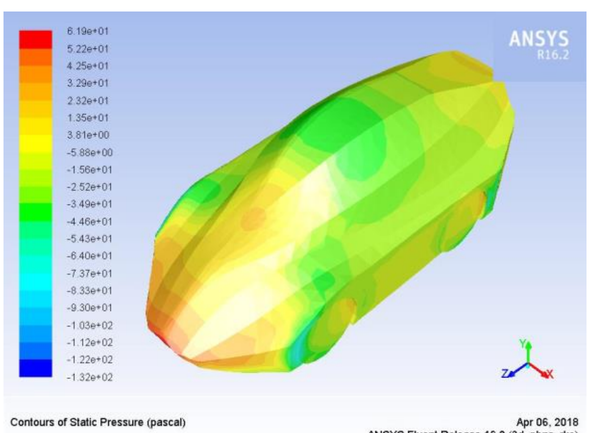
Figure 4.9: Pressure Contour Distribution using ANSYS Fluent 16.2



(d) Model 4

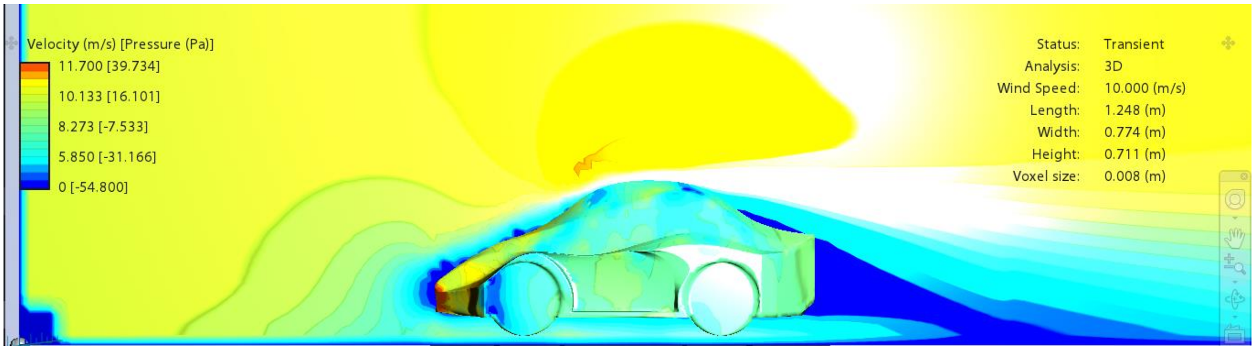


(e) Model 5

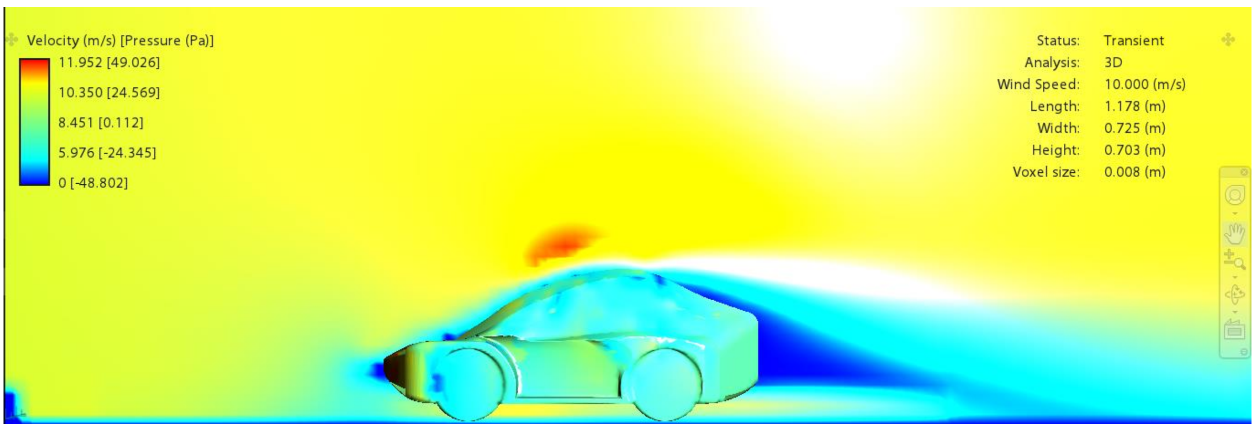


(f) Model 6

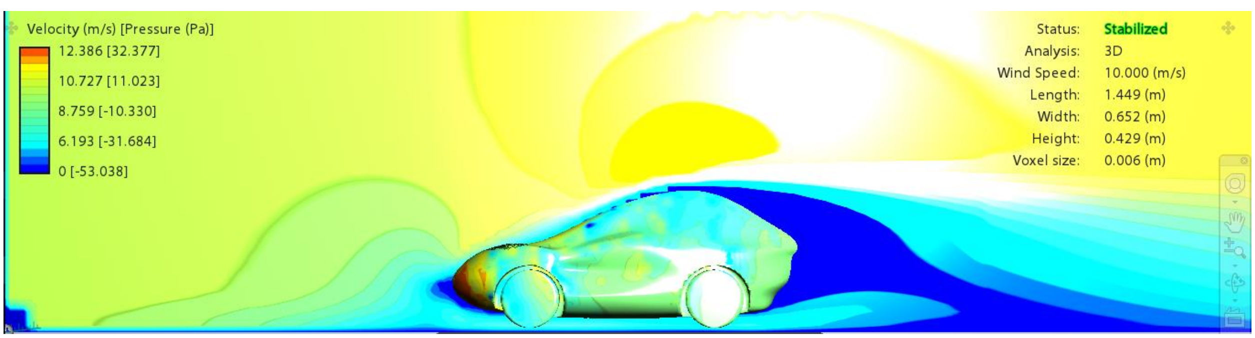
Figure 4.10: Pressure Contour Distribution using ANSYS Fluent 16.2 (cont'd)



(a) Model 1

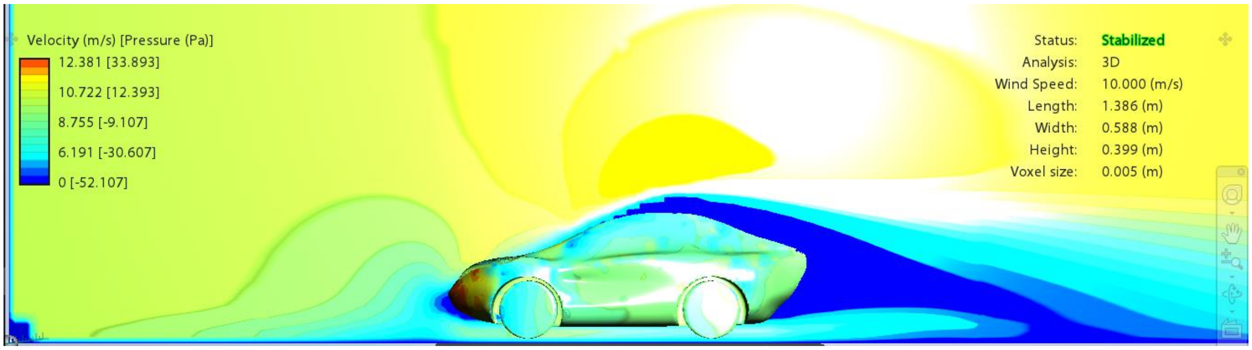


(b) Model 2

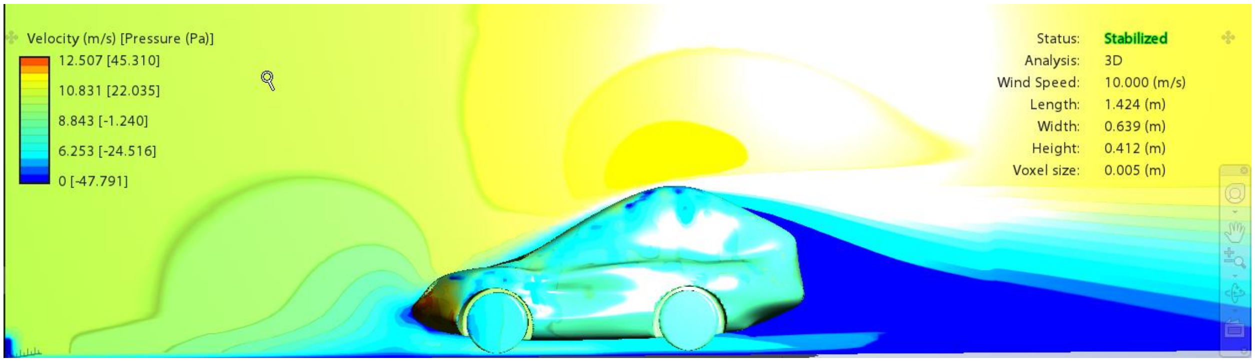


(c) Model 3

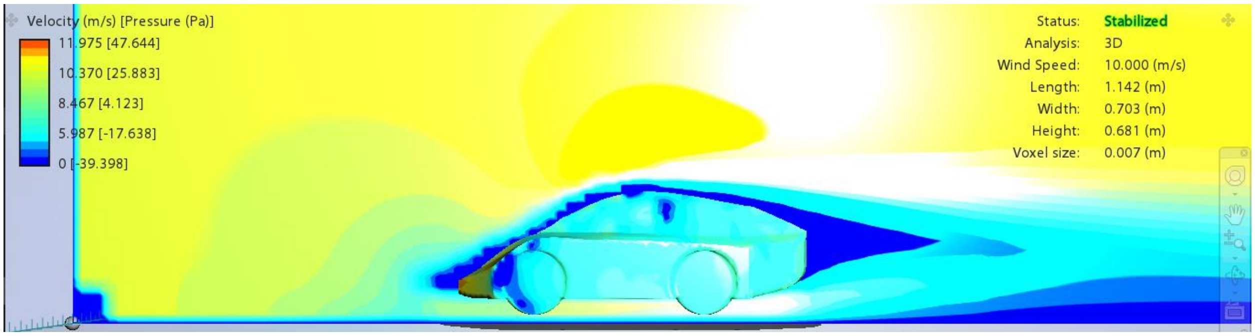
Figure 4.11: Velocity Contour Distribution using Autodesk Flow Design



(d) Model 4

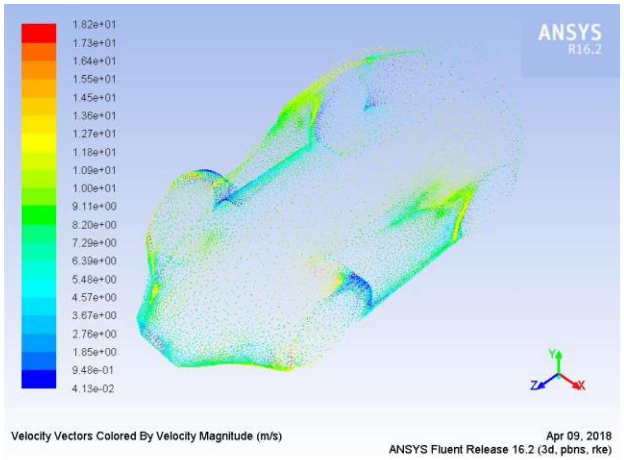


(e) Model 5

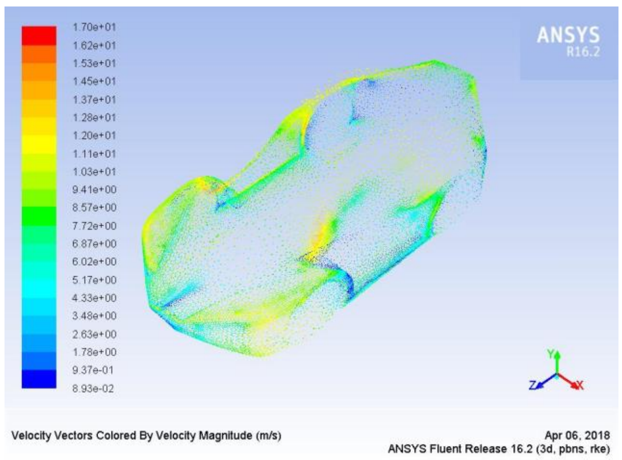


(f) Model 6

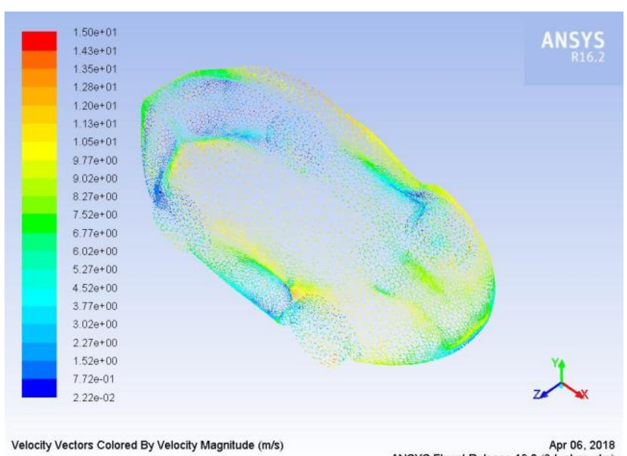
Figure 4.12: Velocity Contour Distribution using Autodesk Flow Design (cont'd)



(a) Model 1

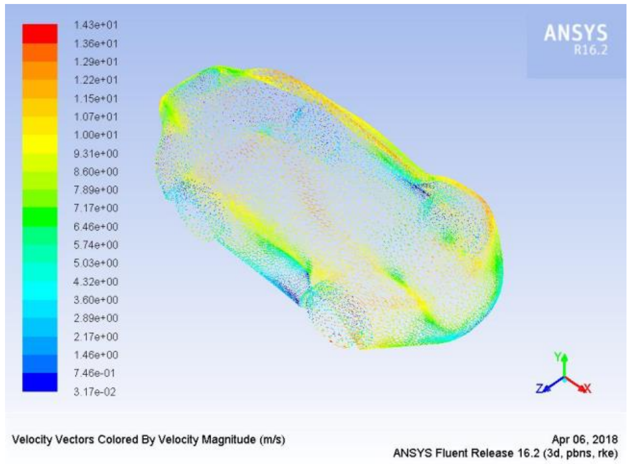


(b) Model 2

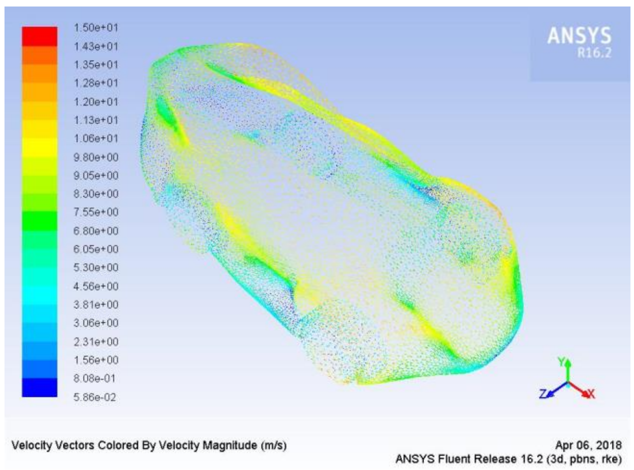


(c) Model 3

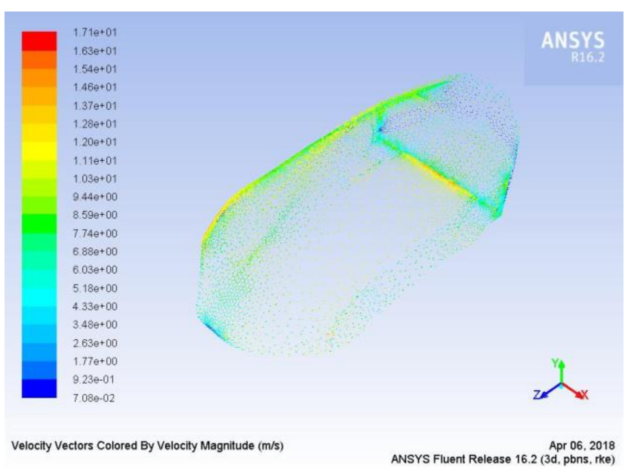
Figure 4.13: Velocity Vector Distribution using ANSYS Fluent 16.2



(d) Model 4



(e) Model 5



(f) Model 6

Figure 4.14: Velocity Vector Distribution using ANSYS Fluent 16.2 (cont’d)

4.3 Evolution of the final concept

Aside the CFD test for the concept models, over 30 simulation experiments were performed to test for the effect of the addition or removal of certain features to the car; this resulted in modification to the concept design for better aerodynamic performance. The results gotten from the experiments are summarized in table 4.2.

The Model6 was therefore ascertained to be the most aerodynamically efficient concept. It was therefore chosen as the concept car to be modeled. The orthographic drawing is shown in Fig. 4.15.

Table 4.2: Drag and Lift Coefficients of the various models at velocity 10 m/s obtained from ANSYS Fluent 16.2

Models	**Drag Coefficient (C_D)**	**Drag Force (F_D)**	**Lift Coefficient (C_L)**	**Lift Force (F_L)**
Model 1	0.42	26.4N	1.17	7.37N
Model 2	0.408	25.0N	0.11	6.8N
Model 3	0.38	23.1N	0.017	1.03N
Model 4	0.35	21.6N	-0.006	-0.35N
Model 5	0.29	17.88N	-0.073	-4.44N
Model 6	0.24	14.8N	-0.38	-23.5N

Table 4.3 Result of the Aerodynamic Tests

Change of body geometry	**Initial Drag (D_o)**	**Final Drag (D_1)**	**Change in Drag ($\Delta D/D_o$x100%)**
Rounding up of trailing and leading edges	0.42	0.408	-1.47%
Rounding up of side edges	0.408	0.38	-6.86%
Streamlining of body width	0.38	0.35	-7.89%
Streamlined tapering of tail part	0.35	0.29	-6.00%
Extending peak height forward	0.29	0.287	-1.03%
Removal of side door intruding feature	0.287	0.24	-16.38%

The overall change in drag = $\frac{0.24-0.42}{0.42} \times 100\% = -42.85\%$

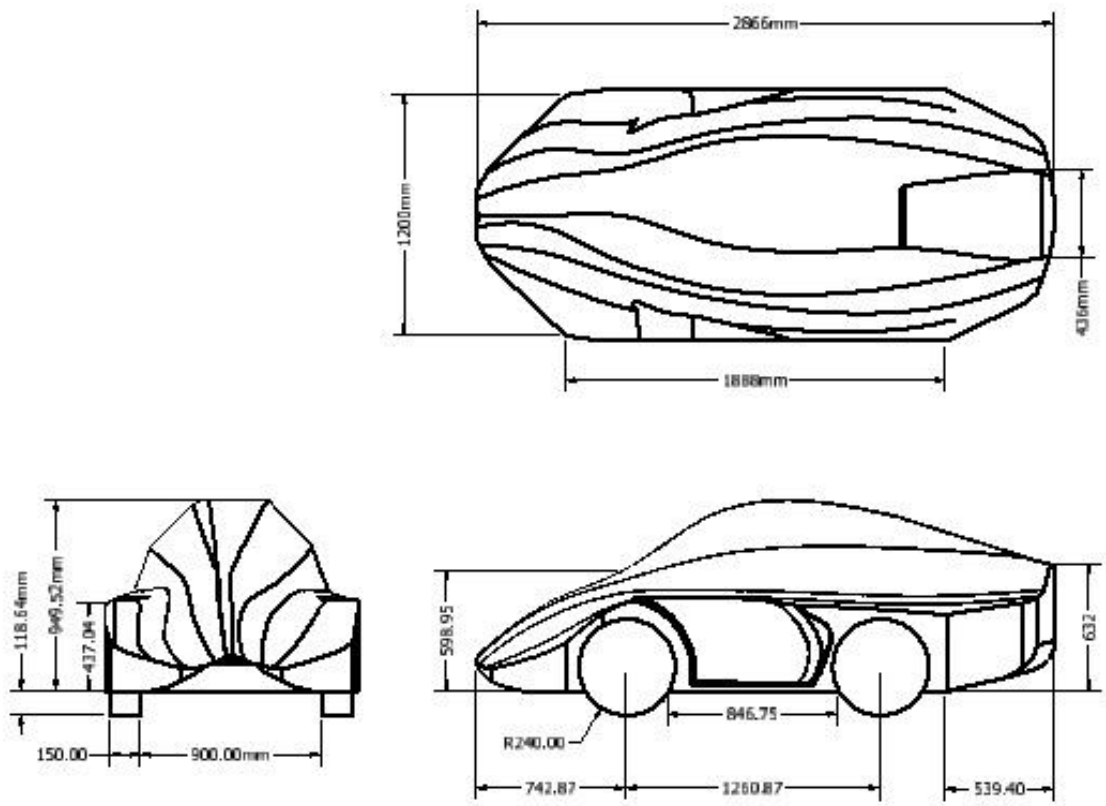


Figure 4.15: Drawing of the Final Concept

4.4 Materials, Tools and Equipment Used for Fabrication Process

4.4.1 Modeling

Tools used include spatula, knife, spray gun, kidney pallet (for smoothening), scrapper, and beater.

Materials used include Clay, metal, wood.

Figure 4.16 to Figure 4.18 shows the car model as it proceeds from the partial execution stage to the detailed execution stage.

4.4.2 Mould making

Materials include Gypsum Plaster (Plaster of Paris), Water.

Tools include Bucket, hand glove, stirrer.

4.4.3 Casting

Materials include Resin (Fiberglass material), Accelerator (for speeding up dryness), catalyst (for speeding up mixture), and Fiber mat (for reinforcement).

Tools include Brush, bucket, and nose guard

4.4.4 Finishing and Partitioning

Materials include Auto-based paint, body filler, and sealant.

Tools include sandpaper

Equipment include angle grinder, spraying machine.

The materials, tools and equipment used in the fabrication process are shown in Figure 4.19

Figure 4.16: Partial execution of the clay model

Figure 4.17: Full execution of clay model

(a) Side View

(b) Front View

Figure 4.18: Detailed execution of clay model

(a) Plaster of Paris (outer)


(b) Fibre Mat


(c) Resin

(d) Spray Painter

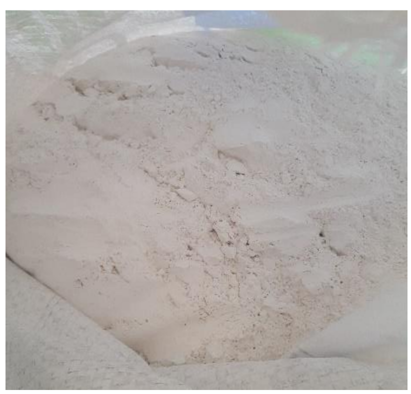

(e) Plaster of Paris (inner)

Figure 4.19: Materials, tools and equipment used during fabrication process

Source: Fine-art Department Studio

Figure 4.20: Completed fabrication of car model

CHAPTER FIVE

CONCLUSION AND RECOMMENDATION

Based on the examined values of the coefficients and forces obtained from the CFD simulation, it can be inferred that varying the geometry of the model significantly causes a change in the values of the resulting coefficients and forces.

As it is used in Automobile industries, CAD and CFD software are also used here to minimize the drag force and increase the down force (negative lift) which helps to stabilize the vehicle, which will lead to decrease in fuel consumption.

The state of different car models in fluid domain can be simulated but different models have different actions in fluid domain and also different method of analysis. The computational fluid dynamics provides visual images and numerical data to simulate the car body in the fluid flow. After the car models were designed using Autodesk Inventor, the CFD analysis was done using ANSYS Fluent, which shows approximate results of real-life wind tunnel tests. The results helped to redesign the shape and geometry in CAD and then re-simulate to get improved results accordingly.

Modification of the body geometry of the initial concept resulted in a reduction of drag coefficient from 0.42 to 0.24, which infers 42.85% reduction in drag, with added improvements in terms of lower noise pollution and lesser mud deposition.

The result also proved that for better aerodynamic design, automotive designers should be more objective than aesthetic in their focus. A streamlined body shape with reduced sharp edges, looking closer to a teardrop shape, is more aerodynamically and thus has greater fuel-efficiency.

Overall, the following remarks can be made about reducing drag:

- Separation zone occurs at the rear of a vehicle and therefore the design of the rear has a very high potential for drag reduction.
- The frontal part of the vehicle is the region where the approaching air tends to be stagnant (reduced velocity), undue protrusion should therefore be avoided and edges should be rounded off as much as possible.
- CFD is a very useful tool in aerodynamic analysis due to their cost effectiveness; information management; and their faster speed when compared to wind tunnel tests.

Recommendations

This concept should be taken up as a student-team work and further developed into a full-scale vehicle for participation in the next Shell Eco-marathon competition. This would expose students to broader practical engineering skills and experience.

Designs and Simulations of the concept car can be done more in-depth to improve the aerodynamic efficiency of the car. Thus, achieve greater prospects for fuel-efficient cars of the nearest future.

REFERENCES

Adeniyi, A.A. (2008). Design of Ultra High Mileage Engine. MSc Thesis, University College London, London, England, UK.

Ahmed F. (1999) Drag Reduction of Pickup Truck using Add-on Deices, Master Thesis, Addis Ababa University , Ethiopia.

Ajao KR, Ajimotokan HA, Popoola OT, Akande HF (2009) Electric energy supply in Nigeria, decentralized energy approach. Cogen Distrib Gen J 24(4):34–50

Akubo, S., Dongo, E.I., Momoh, I.M., Okorie, N.N., Oluyori, R.T., (2013) Revitalization of the Nigerian coal mining industry to expand the power generation needs of Nigeria. J Res Environ Sci Toxicol 2(8):175–178

Bevinkatti, K., Mali, A., Bayas, R., Ghadage, A., Anuse, U. L., Four wheel steering system for Automobile, Novateur Publications International Journal of Innovations in Engineering Research And Technology [IJIERT] ISSN: 2394-3696 VOLUME 2, ISSUE 4APR.-2015

Buresti, G., Lungo, G. V., and Lombardi, G., (2007), "Carried out a Research on Methods for the Drag Reduction of Bluff Bodies and Their Application to Heavy Road".

Collina, T. Z. (2005) Testimony to the United States Senate: "Oil Dependence and U.S. Foreign Policy: Real Dangers, Realistic Solutions." 20/20 Vision.

D'Agostino, A., Bixler, E., Tinholt-Chiza, S., (2013), Final Report: Team 16 Supermileage, Calvin College Engineering, Unpublished B.Sc Thesis.

Da Silva, E.M.; Diwa, J.S.; Pimentel, M.D (2014). ***Design and Simulate an Aeroynamic UrbanConcept Car Body for the Shell Eco-marathon with Less Coefficient of Drag.*** B.Sc Thesis, Mechanical Engineering, Mapua Institute of Technology.

Earleywine, M.; Gonder, J.; Markel, T.; and Thornton, M. 2010. Simulated fuel economy and performance of advanced hybrid electric and plug-in hybrid electric vehicles using inuse travel profiles. Proc. IEEE Vehicle Power and Propulsion Conference (VPPC), Lille, France, 1-3 September 2010. Pp. 1-6.

EIA [U.S. Energy Information Administration]. (2011). International Energy Outlook 2011. Available at: http://www.eia.gov/forecasts/archive/ieo11/

Emodi, N. V., (2016) The Energy Sector in Nigeria, Energy Policies for Sustainable Development Strategies, Frontiers in African Business Research, DOI 10.1007/978 -981 -10-0974-7_2

Energy Commission of Nigeria (ECN) (2007) National energy policy. Federasl Republic of Nigeria, Abuja 2007

Enibe SO, Odukwe AO (1990) Patterns of energy consumption in Nigeria. Energy Convers Manag 30(2):69–73

How A Car Works (2017). "Car Aerodynamics" Retrieved October 2017.

Hui SCM (1997) Renewable energy to sustainability: the challenge for Honk Kong. Hong Kong Institution of Engineers, pp 351–358

Ibitoye FI (2014) Ending natural gas flaring in Nigeria's oil fields. J Sustain Dev 7(3):13

ICCT [International Council on Clean Transportation]. (2014), Global Transportation Roadmap Model. Available at: http://www.theicct.org/global -transportationroadmap-model

Janosko I.; Polonee T.; Kuchar P.; Machal P.; Zach M. (2017). Computer Simulation of Car Aerodynamic Properties. Acta Universitatis Agricultrae et Silviculturae Memdelianae Brunensis. 5(65). 1505-1514.

Khalid, S., Waseem, H., Amjad, H, (2015), Design and Fabrication of a Prototype Car Chassis for Shell Eco-Marathon, Unpublished B. Sc. Thesis, Faculty of Engineering, University of Central Punjab, Lahore Pakistan.

Krishnani P.N. (2006) CFD Study of Drag Reduction of a Generic Sport Utility Vehicle. Masters' Thesis, Mechanical Engineering, University of California, Sacramento.

Lohith, K., Shankapal, S. R., Monish Gowda, M. H., Development of Four Wheel Steering System for a Car, Sastech Journal, Volume 12, Issue 1, April 2013

Michalek J. J., Papalambros P. Y. and Skerlos S. J. (2004), A Study of Fuel Efficiency and Emission Policy Impact on Optimal Vehicle Design Decisions, Journal of Mechanical Design, November 2004, Vol. 126.

Muhamad S. BD., (2012), Design and Analysis of Urban Concept Car Chassis for Shell Eco Marathon Challenge, Unpublished B.Sc. Thesis, Faculty of Mechanical Engineering Universiti Malaysia Pahang.

Nwaobi GC, Nigeria WA, Territory FC (2005) The Nigerian coal corporation: an evaluation of production performance (1960–1987) (No. 0501002). EconWPA

Odesola IF, Eneje S, Olugasa T (2013a) Coal development in Nigeria: prospects and challenges. Int J Eng Appl Sci 4(1):64–73

Odesola IF, Samuel E, Olugasa T (2013b) Coal development in Nigeria: prospects and challenges. Int J Eng 4(1):8269

Oyedepo SO (2012) Energy and sustainable development in Nigeria: the way forward. Energy Sustain Soc 2(1):1–17

Oyedepo SO (2014) Towards achieving energy for sustainable development in Nigeria. Renew Sustain Energy Rev 34:255–272

P. Sainio, K. Killstrom and M. Juhala, (2007), "From Aalto University Conducted a Research on Aerodynamics Possibilities for Heavy Vehicles".

Peterbilt (2009), "Motors Company Presents a White Paper on Heavy Vehicle Aerodynamics and Fuel Efficiency".

Petkar R. K., Kolgiri S. G., and Ragit, S. S., Study of Front-Body of Formula-One Car for Aerodynamics using CFD, (2014), International Journal of Application or Innovation in Engineering & Management (IJAIEM), Volume 3, Issue 3, ISSN 2319 - 4847

Radom, Poland (2008) "Ecology and safety as a driving force in the development of vehicles." ISEP, Porto Portugal

SAE (1998). "Aerodynamics of Road Vehicles", Edited by Wolf-Heinrich Hucho, SAE International, Warrendale, PA, 1998.

Sambo A.S., Iloeje O.C., Ojosu O.J., Olayande S., Yusuf A.O. (2006) Nigeria's experience on the application of IAEA s energy models (MAED & WASP) for national energy planning.

Sapuan S.M; Ham K.W.; Ng K.M; Woo C.K.; Ariffin K.A; Baharudin B.T.H.T; Faleza A.A; Supeni E.E; Jalil N.A.A (2009). "Design of composite racing car body for student based competition." Scientific Research and Essay. Vol. 4(11) pp. 1151-1162

Shell. 2012. History of Shell Eco-marathon. Available: http://www.shell.com/home/content/ecomarathon/about/history

Singh J.; Randhawa J.S. (2014). CFD Analysis of Aerodynamic Drag Reduction of Automobile Car – A Review. International Journal of Science and Research. 3(6)

Sivaraj, G., Gokulraj, M. (2012), Optimum Way to Increase the Fuel Efficiency of the Car Using Base Bleed, International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.3, May-June 2012 pp-1189-1194, ISSN: 2249-6645

Song, E.; Fan, L.; Liu, G.; and Long, W. 2010. Numerical simulation of combination engine HEV on fuel economy. Proc. WASE International Conference on Information Engineering (ICIE), Beidaihe, Hebei, China, 14-15 August 2010. Vol. 4, pp. 244-9.

Steiner, B. and Mauzerall, D. L., (2006), Achieving Vehicle Fuel Efficiency: The CAFE Standards and Beyond.

Sutikno, W., Aulia, D. and Rahman, H., (2016), Modeling of a Shell Eco-Marathon Vehicle Based on Drive-Train Characteristic and Driver Modes to Predict Fuel Consumption of the Vehicle on a Specific Track, ARPN Journal of Engineering and Applied Sciences, VOL. 11, NO.4, FEBRUARY 2016, ISSN 1819-6608.

Turrentine, T.S.; and Kurani, K.S. 2007. Car buyers and fuel economy? Energy Policy: 35(2): 1,213-23.

Wakeham, K.J., (2009), Introduction to Chassis Design, Memorial University of Newfoundland and Labrador

Appendix A: Shell Ecomarathon Rules

3. VEHICLE DESIGN

3A – GENERAL

TECH TIPS

Article 25: **VEHICLE DESIGN**

a) During vehicle design, construction and competition planning, participating Teams must pay particular attention to all aspects of safety, i.e. Driver safety, the safety of other Team members and spectator safety.

   i. Prototype vehicles must have three or four running wheels, which under normal running conditions must be all in continuous contact with the road.

   ii. UrbanConcept vehicles must have exactly four wheels, which under normal running conditions must be all in continuous contact with the road. A fifth wheel for any purpose is forbidden.

b) Aerodynamic appendages, which adjust or are prone to changing shape due to wind whilst the vehicle is in motion, are forbidden.

c) Vehicle bodies must not include any external appendages that might be dangerous to other Team members; e.g. pointed part of the vehicle body. Any sharp points must have a radius of 5 cm or greater, alternatively they should be made of foam or similar deformable material.

d) Vehicle body panels must be rigid with an appropriate stiffness not to be prone to changing shape due to wind.

e) The vehicle interior must not contain any objects that might injure the Driver during a collision.

f) Windows must not be made of any material which may shatter into sharp shards. Recommended material: Polycarbonate (e.g. Lexan)

g) Any cover of the energy compartment (engine/motor/transmission/battery, etc.) should be easy to open for quick inspection access.

h) All parts of the drive train, including fuel tank, hydrogen system components, etc. must be within the confines of the body cover.

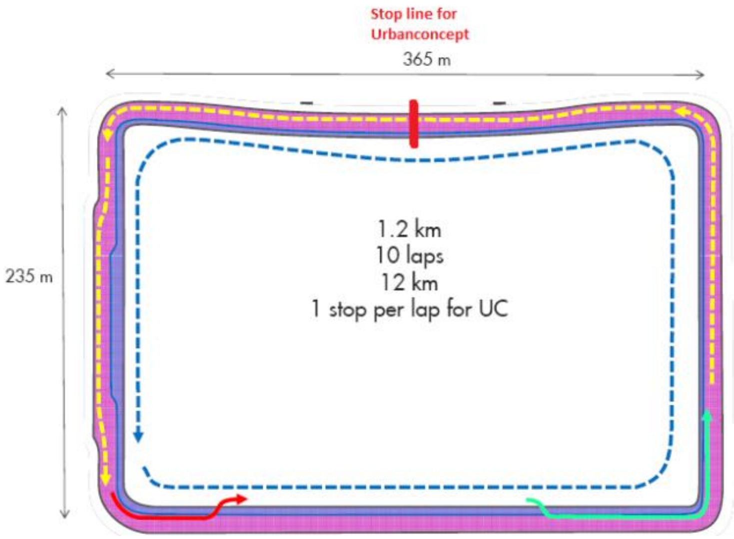
i) All objects in the vehicle must be securely mounted. Bungee cords or other elastic material are not permitted for securing heavy objects like batteries.

j) All vehicles must have a solid floor and frame that prevents any part of the driver's body from contacting the ground.

k) All vehicles (including Prototypes) must be fully covered. Open top vehicles are not allowed. Vehicles that look like bicycles, tricycles or wheelchairs are not acceptable.

l) The Organisers may provide any team with telemetry equipment and request them to install it in their vehicle for the purpose of competition monitoring and result calculation. In this case the main housing of the telemetry equipment will need to be installed inside the vehicle and the team must provide a hole in the body of the vehicle of no more than15 mm for the passage of cables to one or more outside antennae which will need to be attached outside on top of the vehicle.

Appendix B: Computations for Wind Speed



A typical Shell Eco-Marathon Track
Source: (DaSilva et.al, 2014)

Stop time = 2 sec (at least); say 5 sec stop time per lap.
Total stop time, t_{stop}-total = 5 sec x 10 laps = 50 sec; say 1 min per trial
Total time to finish, t_{total} = 29 min (includes stopping time)
Max time for car to finish at track, t_{max} = 29min – 1 min = 28 min

In compliance with the Shell Eco-marathon regulations, DaSilva estimated the minimum speed that would be required to finish a race within the allowed lap time to be 26 km/h; while the maximum speed allowed on the race track is 50km/h. The mean speed is therefore calculated thus:

$$V_{mean} = \frac{50\text{km/h} + 26\text{ km/h}}{2} = 38km/h$$

$$V_{mean} = 38km/h = 38\frac{km}{h} \times \frac{1000m}{1km} \times \frac{1h}{3600s} = 10.56m/s$$

With the assumption that the wind speed while the car is static is relatively low and thus negligible to the average speed of the moving car, the car's average speed while in motion is taken to be the approximate wind speed in our simulation.

$$\therefore\ Vwind \cong 10m/s$$

Appendix C: Rendered Design of the Urban-Concept Vehicle