DESIGN, ANALYSIS AND OPTIMIZATION OF A FORMULA STYLE RACING CAR USING COMPUTATIONAL FLUID DYNAMICS

A thesis submitted in partial fulfillment of the requirements for the Degree of
Bachelor of Technology
Automotive Design Engineering

By
Paras Kanwar
Prateek Kumar
Sandeep Singh Pannu

Under the guidance of

Mr. Deepak Kumar
Assistant Professor, University of Petroleum & Energy Studies

Approved

Dr. Shrihari
Dean

College of Engineering
University of Petroleum & Energy Studies
Dehradun
May, 2010

CERTIFICATE

This is to certify that the work contained in this thesis titled "*DESIGN, ANALYSIS AND OPTIMIZATION OF A FORMULA STYLE RACING CAR USING COMPUTATIONAL FLUID DYNAMICS*" has been carried out by Paras Kanwar, Prateek Kumar and Sandeep Pannu under my supervision and has not been submitted elsewhere for a degree. Their work is original and genuine to the best of our knowledge.

Deepak Kumar

Mr. Deepak Kumar
Assistant Professor,
University of Petroleum & Energy Studies

Abstract

This project uses Computational Fluid Dynamics to simulate for various aerodynamics parameters such as lift and drag forces. Initially the frame of a formula style vehicle was designed using Ergonomics and Society of Automotive Engineers Formula SAE guidelines and analyzed by Finite Element Method for stress. A 3-d model of 95th Percentile male was created and the vehicle designed using it as a template. Various body configurations were made and analyzed on Ansys CFX with the aim of reducing drag and generating as much negative lift as possible. A final optimized model was created after a Drag/Downforce compromise with a very high downforce with minimal increment in the drag force.

ACKNOWLEDGEMENT

I am thankful to **Dr. Mukesh Saxena,** Head of department Automotive Design Engineering, college of engineering studies, University of Petroleum and Energy Studies for explaining us the various projects that we could undertake and assigned **Mr. Deepak Kumar**, Assistant Professor, College of engineering studies, University of Petroleum and Energy Studies as our mentor.

I express deep gratitude towards **Mr. Deepak Kumar** and **Dr. Ugur Guven** who explained us the various work and tests that were of importance to our project, amidst his busy schedule.

Further, I am grateful to the library staff for their valuable service without which this project would not have been completed.

I am thankful to our dean **Dr. Srihari** who was a great inspiration for us throughout the project.

Finally I express a very special sense of appreciation to my parents and the entire University of Petroleum and Energy Studies family who benevolently supported me during my course.

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

The objective of a race car is to win races- whether one sees racing as a sport, a field of new technology or a design and R&D field. Motor sport is very complex and he job of the people who sit down and design and the concept have a lot of work on their hands. At the very beginning of this work comes the problem of performance from the driver vehicle which in he given environment beats the competition. The dynamic behavior of the high tech machines and the enormously complex human beings that make this so much more than just a motor sport.

The vehicle can be modified and adjusted to enhance the driver capabilities and facilitate high driver control, which is known as vehicle dynamics. The performance of the vehicle and the way the driver drives it is directly proportional. The high speed sports cars are like Ferraris, Lamborghinis, TVRs, bugattis and many others. They are high performance and highly priced vehicles with all new tech.

We have devised a design similar to a formula 1 vehicle which goes a top speed of 320 kmph and has a thousand horsepower. The aerodynamic study of this kind of vehicle is the most important as the forces of air on such high speeds play an important part to understand the stability and ease of driving that vehicle. The pproject revolves all around from the basics of aerodynamic to the most complex CFD analysis on ANSYS.

CHAPTER 2

LITERATURE REVIEW

1.1 SAE GUIDELINES taken into account:

FORMULA SAE INTERNATIONAL RULEBOOK 2009:

2.1.1 General Design Requirements

2.1.1.1 Body and Styling

The vehicle must be open-wheeled and open-cockpit (a formula style body). There must be no openings through the bodywork into the driver compartment from the front of the vehicle back to the roll bar main hoop or firewall other than that required for the cockpit opening. Minimal openings around the front suspension components are allowed.

2.1.1.2 Wheelbase and Vehicle Configuration

The car must have a wheelbase of at least 1525 mm (60 inches). The wheelbase is measured from the center of ground contact of the front and rear tires with the wheels pointed straight ahead. The vehicle must have four (4) wheels that are not in a straight line.

2.1.1.3 Vehicle Track

The smaller track of the vehicle (front or rear) must be no less than 75% of the larger track.

2.1.2 Chassis Rules

2.1.2.1 Ground Clearance

Ground clearance must be sufficient to prevent any portion of the car (other than tires) from touching the ground during track events.

2.1.3 Structural Requirements

Among other requirements, the vehicle's structure must include two roll hoops that are braced, a front bulkhead with support system and Impact Attenuator, and side impact structures.

2.1.3.1 Definitions

The following definitions apply throughout the Rules document:

(A) Main Hoop - A roll bar located alongside or just behind the driver's torso.

(B) Front Hoop - A roll bar located above the driver's legs, in proximity to the steering wheel.

(C) Frame Member - A minimum representative single piece of uncut, continuous tubing.

(D) Frame - The "Frame" is the fabricated structural assembly that supports all functional vehicle systems. This assembly may be a single welded structure, multiple welded structures or a combination of composite and welded structures.

(E) Primary Structure – The Primary Structure is comprised of the following Frame components: 1) Main Hoop, 2) Front Hoop, 3) Roll Hoop Braces, 4) Side Impact Structure, 5) Front Bulkhead, 6) Front Bulkhead Support System and 7) all Frame Members, guides and supports that transfer load from the Driver's Restraint System into items 1 through 6.

(F) Major Structure of the Frame – The portion of the Frame that lies within the envelope defined by the Primary Structure. The upper portion of the Main Hoop and the Main Hoop braces are not included in defining this envelope.

(G) Front Bulkhead – A planar structure that defines the forward plane of the Major Structure of the Frame and functions to provide protection for the driver's feet.

(H) Impact Attenuator – A deformable, energy absorbing device located forward of the Front Bulkhead.

2.1.3.3 Minimum Material Requirements

2.1.3.3.1 Baseline Steel Material

The Primary Structure of the car must be constructed of:

Either: Round, mild or alloy, steel tubing (minimum 0.1% carbon) of the minimum dimensions specified in the following table:

ITEM or APPLICATION	OUTSIDE DIAMETER x WALL THICKNESS
Main & Front Hoops	1.0 inch (25.4 mm) x 0.095 inch (2.4 mm) or 25.0 mm x 2.50 mm metric
Side Impact Structure. Front Bulkhead. Roll Hoop Bracing & Driver's Restraint Harness Attachment	1.0 inch (25.4 mm) x 0.065 inch (1.65 mm) or 25.0 mm x 1.75 mm metric or 25.4 mm x 1.60 mm metric
Front Bulkhead Support	1.0 inch (25.4 mm) x 0.049 inch (1.25 mm) or 25.4 mm x 1.25 mm metric

TABLE 2.1 : Minimum dimensions

2.1.3.4 Roll Hoops

The driver's head and hands must not contact the ground in any rollover attitude.

The Frame must include both a Main Hoop and a Front Hoop as shown in Figure 2.1.

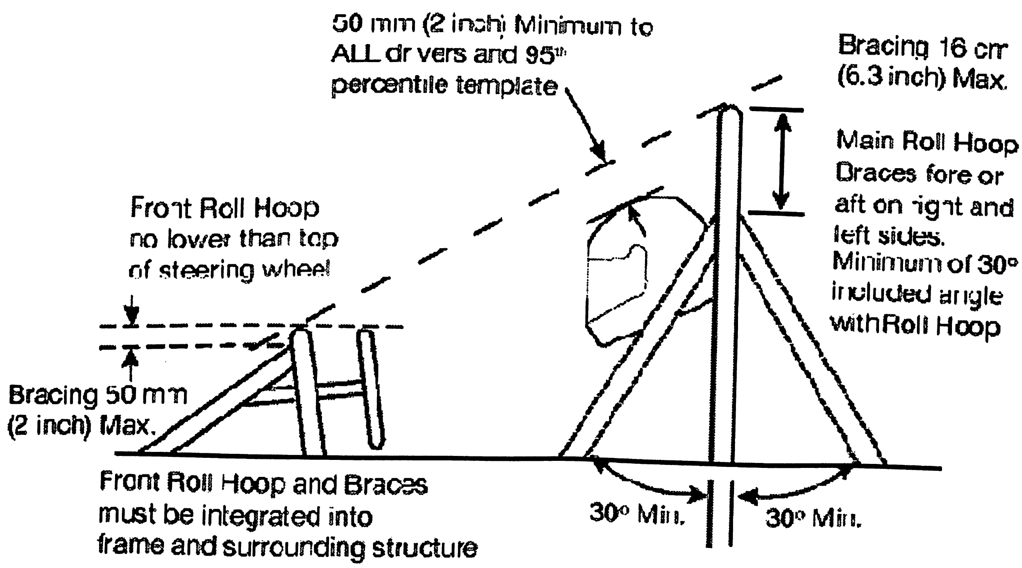


FIGURE 1

Fig 1: Side view dimensions

2.1.3.4.1 Main and Front Hoops – General Requirements

(A) When seated normally and restrained by the Driver's Restraint System, a straight line drawn from the top of the main hoop to the top of the front hoop must clear by 50.8 mm (2 inches) the helmet of all the team's drivers and the helmet of a 95th percentile male (anthropometrical data).

(B) The minimum radius of any bend, measured at the tube centerline, must be at least three times the tube outside diameter. Bends must be smooth and continuous with no evidence of crimping or wall failure.

(C) The Main Hoop and Front Hoop must be securely integrated into the Primary Structure using gussets and/or tube triangulation.

2.1.3.4.2 Main Hoop

(A) The Main Hoop must be constructed of a single piece of uncut, continuous, closed section steel tubing per Section 3.3.3.

(B) The use of aluminum alloys, titanium alloys or composite materials for the Main Hoop is prohibited.

(C) The Main Hoop must extend from the lowest Frame Member on one side of the Frame, up, over and down the lowest Frame Member on the other side of the Frame.

(D) In the side view of the vehicle, the portion of the Main Roll Hoop that lies above its attachment point to the Major Structure of the Frame must be within 10 degrees of the vertical.

(E) In the front view of the vehicle, the vertical members of the Main Hoop must be at least 380 mm (15 inch) apart (inside dimension) at the location where the Main Hoop is attached to the Major Structure of the Frame.

2.1.3.4.3 Front Hoop

(A) The Front Hoop must be constructed of closed section metal tubing per Section 1.3.3.

(B) The use of composite materials is prohibited for the Front Hoop.

(C) The Front Hoop must extend from the lowest Frame Member on one side of the Frame, up, over and down to the lowest Frame Member on the other side of the Frame. With proper gusseting and/or triangulation, it is permissible to fabricate the Front Hoop from more than one piece of tubing.

(D) The top-most surface of the Front Hoop must be no lower than the top of the steering wheel in any angular position.

(E) In side view, no part of the Front Hoop can be inclined at more than twenty (20) degrees from the vertical.

2.1.3.6.1 Bulkhead

(A) The Front Bulkhead must be constructed of closed section tubing per Section 2.3.3.

(B) The Front Bulkhead must be located forward of all non-crushable objects, e.g. batteries, master cylinders.

(C) The Front Bulkhead must be located such that the soles of the driver's feet, when touching but not applying the pedals, are rearward of the bulkhead plane. (This plane it defined by the forward-most surface of the tubing.) Adjustable pedals must be in the forward most position.

2.1.3.8.1 Tube Frames

The Side Impact Structure must be comprised of at least three (3) tubular members located on each side of the driver while seated in the normal driving position, as shown in Figure 5. The three (3) required tubular members must be constructed of material per Section 3.3.3.(SAE rule book). The locations for the three (3) required tubular members are as follows:

(A) The upper Side Impact Structural member must connect the Main Hoop and the Front Hoop at a height between 300 mm (11.8 inch) and 350 mm (13.8 inch) above the ground with a 77kg (170 pound) driver seated in the normal driving position. The upper frame rail may be used as this member if it meets the height, diameter and thickness requirements.

(B) The lower Side Impact Structural member must connect the bottom of the Main Hoop and the bottom of the Front Hoop. The lower frame rail/frame member may be this member if it meets the diameter and wall thickness requirements.

(C) The diagonal Side Impact Structural member must connect the upper and lower Side Impact Structural members forward of the Main Hoop and rearward of the Front Hoop. With proper gusseting and/or triangulation, it is permissible to fabricate the Side Impact Structural members from more than one piece of tubing.

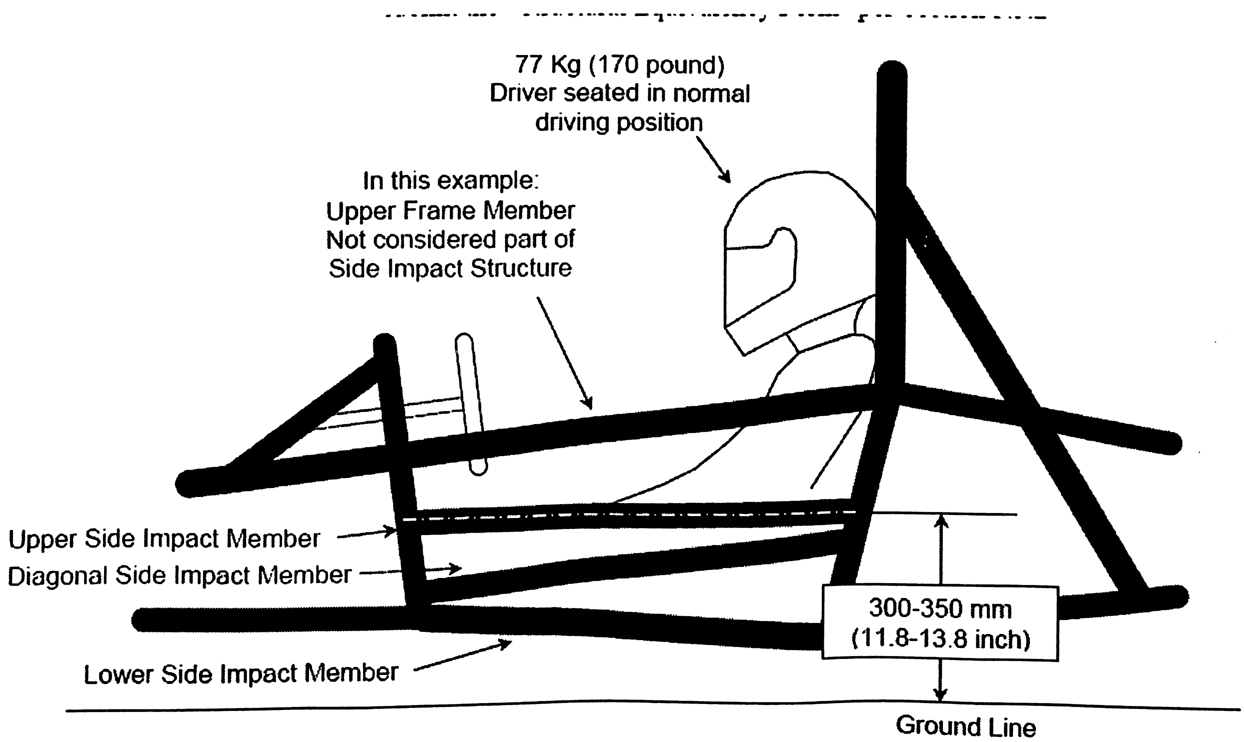


Fig 2: Location of SIM, Diagonal SIM.

2.2. ERGONOMIC ASPECTS:

Ergonomics is the study of the interaction between people and machines and the factors that affect the interaction. Its purpose is to improve the performance of systems by improving human machine interaction. This can be done by 'designing-in' a better interface or by 'designing-out' factors in the work environment, in the task or in the organization of work that degrade human–machine performance.

Systems can be improved by

• Designing the user-interface to make it more compatible with the task and the user. This makes it easier to use and more resistant to errors that people are known to make.

• Changing the work environment to make it safer and more appropriate for then task.

• Changing the task to make it more compatible with user characteristics.

• Changing the way work is organized to accommodate people's psychological, and social needs. The focus is on the interaction between the person and the vehicle and the design of the interface between the two.

The vehicle was to be designed for a 95th percentile male American and the data of human body measurements were taken into account:

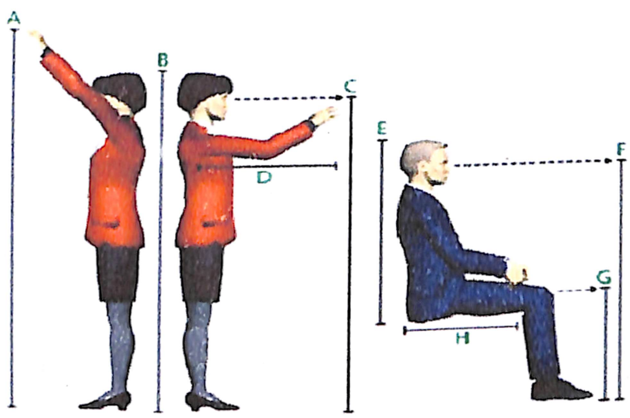


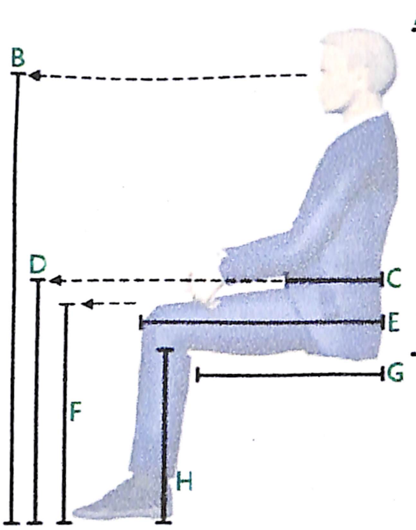
Fig. 3: Human templates

Measurement	Letter	Female	Male
Standing Overhead Reach	A	74.9" – 86.8"	81.2" – 93.7"
Standing Height	B	60.2" – 68.4"	64.8" – 73.5"
Standing Eye Height	C	56.9" – 65.0"	61.4" – 69.8"
Standing Forward Reach	D	30.8" – 36.1"	33.8" – 39.5"
Sitting Height	E	31.3" – 35.8"	33.6" – 38.3"
Sitting Eye Height	F	42.6" – 48.8"	46.3" – 52.6"
Sitting Knee Height	G	19.8" – 23.2"	21.4" – 25.0"
Seat Depth	H	16.9" – 20.4"	17.7" – 21.1"

Table 2.2: Anthropometric Data (US Military)

2.1 Anthropometric Databases

Anthropometric datasets compare people of different ages and occupations. Data in anthropometric databases may represent static dimensions, such as "lower leg length" or functional dimensions such as "reach."



Measurement	Letter	Female 5th – 95th%	Male 5th – 95th%	Overall Range 5th – 95th%
Sitting Height	A	31.3" – 35.8"	33.6" – 38.3"	31.3" – 38.3"
Sitting Eye Height	B	42.6" – 48.8"	46.3" – 52.6"	42.6" – 52.6"
Waist Depth	C	7.3" – 10.7"	7.8" – 11.4"	7.3" – 11.4"
Thigh Clearance	D	21.0" – 24.5"	23.0" – 26.8"	21.0" – 26.8"
Buttock-to-Knee	E	21.3" – 25.2"	22.4" – 26.3"	21.3 – 26.3"
Knee Height	F	19.8" – 23.2"	21.4" – 25.0"	19.8" – 28.0"
Seat Length/Depth	G	16.9" – 20.4"	17.7" – 21.1"	16.9" – 21.1"
Popliteal Height	H	15.0" – 18.1"	16.7" – 19.9"	15.0" – 19.9"
Seat Width	Not Shown	14.5" – 18.0"	13.9" – 17.2"	13.9" – 18.0"

Table 2.3: Anthropometric measurements (2)

Anthropometric measurements (including allowances for clothing) of small and large males and females, from BIFMA Ergonomics Guidelines, 2002. All measurements are in inches.

2.3 BASIC VEHICLE AERODYNAMICS:

Both aerodynamic forces and the tyre forces affect the maneuvering performance. Unlike the tyre forces which are independent to speed the aerodynamic drag increases rapidly with increase in the vehicles velocity. The aerodynamic drag on a vehicle determines many things about it like the maximum speed, the forward acceleration and the braking/deceleration. The negative lift (down force) increases the load on the wheels but in turn increases the lateral force bearing capacity.

The aerodynamic specialist is confronted with many problems like:

1. Reducing the aerodynamic forces by external shaping, flow properties and design for internal flow.
2. Generating as much down force on the vehicle.
3. Wind tunnel testing, interpretation of the tunnel testing and full scale aerodynamic testing.

2.3.1 PROPERTIES OF AIR:

2.3.1 PROPERTIES OF AIR:

The interaction between the particles of a gas mixture, in this case air, is of two kinds: Two particles can collide with each other with no sliding at the interface (producing "normal" or pressure forces) or two particles can slide by each other (producing "tangential" or shear forces). These interactions can be visualized if the air particles were cubes, as sketched below.

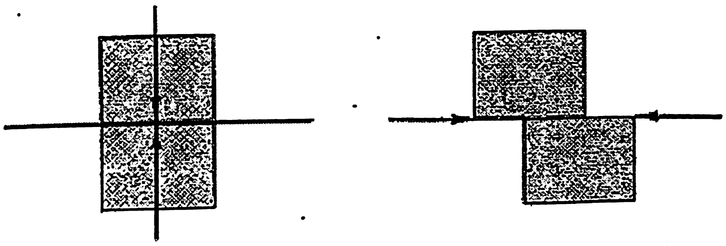


Figure 31: Interaction of air

In reality, both types of interactions occur simultaneously but the relative magnitude of the two types of forces varies widely.

- In most of the air flowing around a body the tangential and shear forces are very small in comparison to the normal forces as the air particles interact with forces perpendicular to the line of contact.
- In the thin layer of air next to a body, called the boundary layer, the shear forces are considered. The layer of air adheres to the body and the particles of air in short distance from the body are at free stream velocity.
- The is an additional situation in which an interaction occurs between the boundary layer and the free stream which can lead to so called separated flow. This turbulent and stalled condition generates high drag and loss of lift and leads to high degree of rotational forces.

2.3.2 BERNOULLI'S EQUATION:

This equation describes the conditions existing in the free stream, outside the boundary layer. It deals with the pressure (normal forces) and the velocity as air flows smoothly around the body. It is possible to predict surface pressures and lift for many types of shapes, especially where no separation is present and boundary layer is small. Drag is not related to this equation.
The flow outside the boundary layer is non viscous with normal or pressure forces between the particles. Considering a race car with subsonic speeds the density change around the body is very small.
An assumption is made that for smooth airflow outside the boundary layer is incompressible and non viscous. These two assumptions define a "perfect fluid". An example of this kind of fluid is water. It's incompressible and with hardly any internal viscosity.
This equation gives us a means of relating velocity and pressure change. It is based on a perfect fluid and is really a statement of conservation of energy in a flow system.

Consider a tube of air, a stream tube , made up of streamlines.

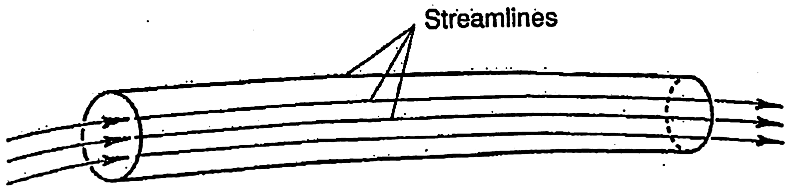


Figure 32: stream tube

A streamline is the path of successive air particles in a steady flow. The outer surface of this tube is in effect of a boundry since no air flows through it. The size and velocity of the tube will be described by bernoulli's equation. Such tubes of air might, for example, pass over and under a wing. The cross-sectional area of these tubes will change with pressure changes as if they were elastic.

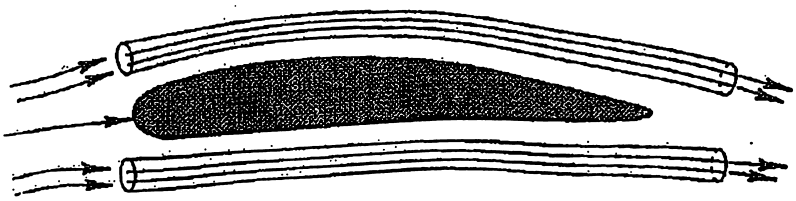


Figure 33: Effect of boundary on flow

The conservation of energy in this flow system is given by the sum of the kinetic energy and the potential energy.

- Potential energy is associated with a height change of the fluid. It is given by Wz, where W is the specific weight and z is the height change. For automotive flow, this term is negligible.

- Kinetic energy is given by $(1/2)mV^2$, where m is the mass (in slugs) and V is the velocity in ft./sec. consider a small volume (vol) of fluid in the sream tube of weight, W, and velocity, V, as shown below:

Considering a small volume in the streamline tube

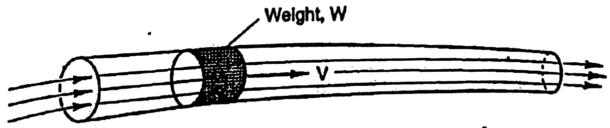


Figure 34: small volume in a streamline tube.

Bernoulli's equation says that this energy is constant along the stream tube, thus,

$$P + (\tfrac{1}{2}\rho V^2) = \text{constant} = H$$

Or

$$P + q = H, \text{ Bernoulli's equation}$$

Where $q = \tfrac{1}{2}\rho V^2$

2.3.3 USE OF BERNOULLI'S EQUATION:

1. To measure the static pressure in the free stream. It is measured by a barometer or by some kind of pressure gauge. The static pressure of a free stream is measured by using a static tube.

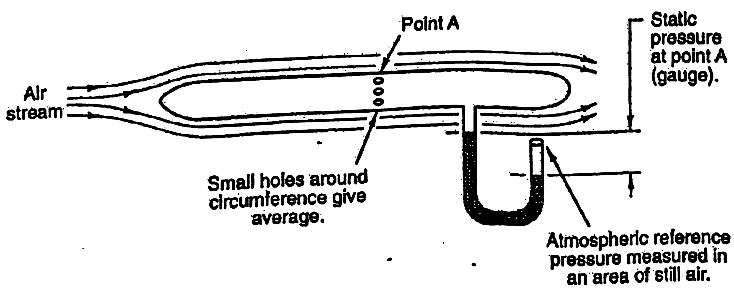


Figure 35: Static pressure in free stream

2. To measure the static pressure along a surface, the flow should be moving parallel to the surface. The principal requirement is that the there should be no component of flow velocity into static hole.

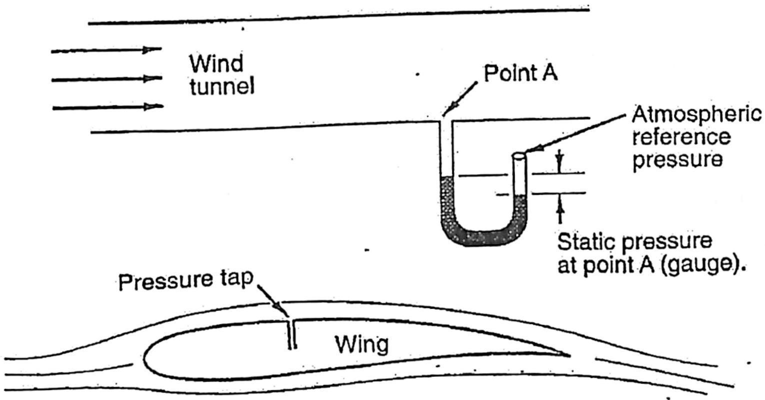


Figure 36: Static pressure along a surface

3. Dynamic pressure: q is the dynamic pressure which is that kinetic energy of the flow system at any given point is converted into pressure. The pressure generated is called stop pressure and are numerically equal. If a flat plat is inserted perpendicular to the direction of flow the air particles in the central stream will lose all its KE and be converted into stop pressure.

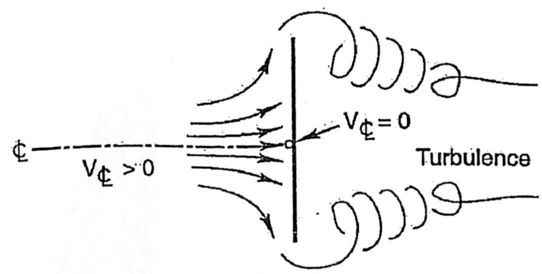


Figure 37: Dynamic pressure

2.3.4 PRESSURE DISTRIBUTON:

The equation states that if the dynamic pressure increases the static pressure will reduce. An airfoil is that device that utilizes the pressure changes to generate forces. The air speed will increase when it goes over both top and bottom surfaces of the airfoil, but the speed is more when it travels over the top surface as distance and curvature is much greater that the bottom surface.

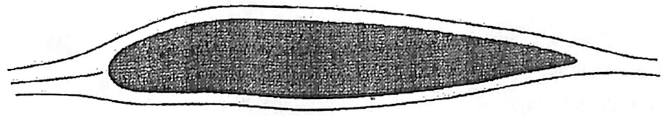


Figure 38: airfoil

As the fluid is incompressible the equation of continuity has to be kept in mind, which is the number of particles at the leading edge for a given volume should be same as the particle at the trailing edge. The stream tubes will be thinner where the velocity of air is higher, over the thicker part of the air foil. The basic stream tubes over and below an airfoil.

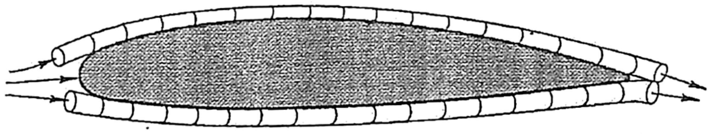


Figure 39: Stream tubes in an airfoil

Lift is generated on this airfoil as the reduction in pressure on the top surface is much greater than the reduction in pressure on the bottom surface, due to velocity being higher on top. If the reduction on pressure and the free stream pressure is plotted on a graph it would look like this.

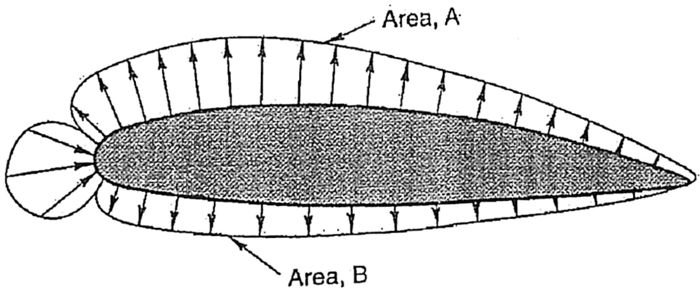


Figure 40: upward lift in airfoil

The pressure distribution in Area A is much greater than the distribution in area B, resulting in upward lift.. The arrows define both pressure and direction, being all normal to d direction of travel of flow.

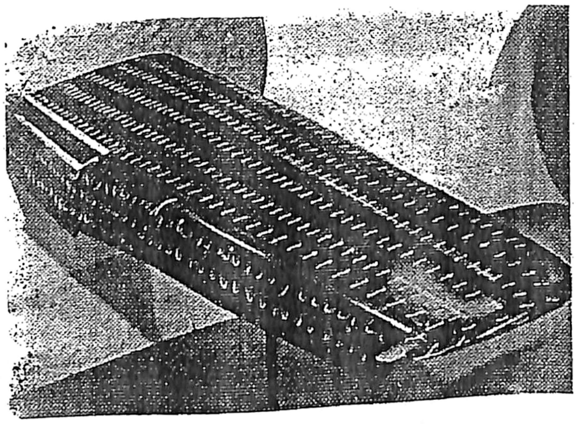


Fig. 4: Actual wing as an airfoil

2.3.5 AERODYNAMIC FORCE/MOMENT COEFFICIENTS:

These forces and moments are primarily due to the pressure changes over and below the body of consideration. These pressure changes vary directly with the dynamic pressure change of the free stream flow system. The actual forces are also proportional to the area on which these pressure changes act on. It is difficult to calculate the area always. A coefficient must be introduced to take into account the effect of body shapes on the velocity distribution over the body and the orientation of the body in that fluid. The coefficient called the drag coefficient (Cd) and defined as:

$$Cd = Drag \div A.q = D/(A.q) = (D/A) \div q$$

Where:

D= drag in lb
A= reference are in ft^2, usually the frontal area of the vehicle
q= dynamic pressure in lb/ft^2
Cd= this is non dimensional

CHAPTER 3

THORETICAL DEVELOPMENT

3.1 **Ergonomic studies and modeling of a 3-D Real life Mannequin**

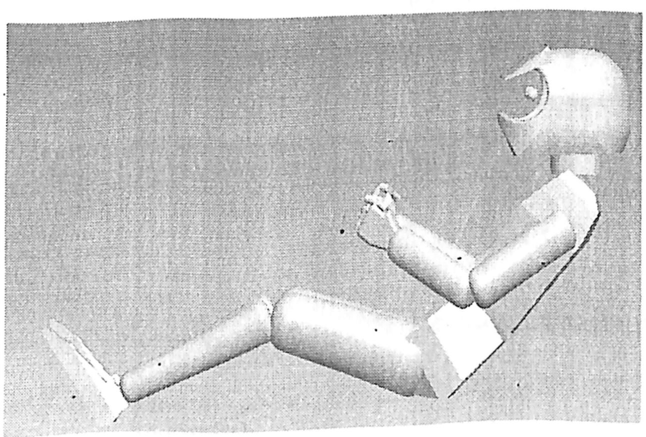
3.2 Anthropometric data for 95^{th} percentile male was taken (tables 2.2 and 2.3), and 3-D model was made of the male in a comfortable driving position. It was also studied that what postures would be most favorable for the long driving conditions. Effect of fatigue and stress on the muscles was also a part of our studies, with sources stating effects ranging from short duration aches to long term migraines.

3.3 Data taken into account:

3.4 Angle between the arms and the fore-arms while clutching the steering: 120 degrees

Angle between the seat horizontal and the lumbar support: 70 degrees

Angle between the thighs and the legs: 136 degrees



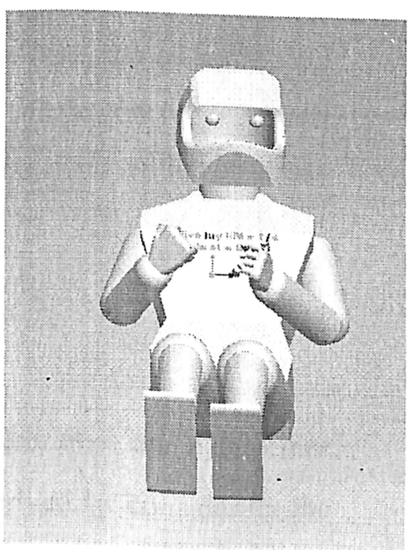


Fig. 5: The 3-D Mannequin

The ergonomics had to be considered as driving is an important and one of the most crucial part of the race winning strategy and the driver is to sit in the cockpit for hours in the cramped position which could easily induce body aches and stress.

The 3-D mannequin was drafted on the software PRO/Engineer by modeling of different parts such as torso, Fingers, arms etc differently and the assembled so that this model could be used in different postures, from standing to lying down.

3.2 USE OF COMPUTATIONAL FLUID DYNAMICS:

In our simulation, using numerical methods, finite volume method, a full model of a formula body was developed and simulated. A hybrid mesh for **turbulent, viscid, steady and transient state, 3d simulation** was used. Surface as well as Volumetric meshes were created.

Here a finite computational fluid domain was made in which the vehicle bluff body was incorporated. The surface mesh of the body was created and then the domain was discretized into fine elements using tetrahedral elements. Element control was given by varying element size, radius of influence, inflations, maximum and minimum spacing. The next step was to give the boundary conditions after defining regions. The boundary conditions remained same for all the analyzed bodies. The various equations used are:

3.2.1. Continuity equation

A continuity equation is a differential equation that describes the transport of some kind of conserved quantity. Since mass, energy, momentum, electric charge and other natural quantities are conserved, a vast variety of physics may be described with continuity equations.

The general form for a continuity equation is

$$\frac{\partial \varphi}{\partial t} + \nabla . v = s$$

Where φ is some quantity, v is a vector function describing the flux of φ, and s describes the generation (or removal) rate of φ. This equation may be derived by considering the fluxes into an infinitesimal box. If φ is a conserved quantity, the generation or removal rate is zero:

$$\frac{\partial \varphi}{\partial t} + \nabla . v = 0$$

In fluid dynamics, the continuity equation is a mathematical statement that, in any steady state process, the rate at which mass enters a system is equal to the rate at which mass leaves the system. The differential form of the continuity equation is:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = 0$$

Where ρ is fluid density, t is time, and u is fluid velocity. If density (ρ) is a constant, as in the case of incompressible flow, the mass continuity equation simplifies to a volume continuity equation:

$$\nabla.u = 0$$

this means that the divergence of velocity field is zero everywhere.

3.2.2. Momentum equation:

The momentum equation is based on Newton's second law of motion that states that the net force on the fluid element equals its mass times the acceleration of the element.

Thus the x component of the momentum equation is given by:

$$\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

Similarly the y and z components are given by:

$$\frac{\partial(\rho v)}{\partial t} + \nabla.(\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla.(\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_z$$

3.2.3. Energy equation:

The energy equation is based on the physical principle that energy is conserved. It can neither be created nor destroyed. It can only be transformed from one form to another.

Thus the energy equation is given by:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right] + \nabla.\left[\rho\left(e+\frac{V^2}{2}\right)V\right] = \rho\dot{q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) - \frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z}$$
$$+ \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} + \rho f.V.$$

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number.

Various turbulence models are used in CFD, such as the Reynolds Stress Model, k-Epsilon model; K- Omega etc are used. The turbulence model used by us it the Shear Stress Transport model which is used to accurately simulate boundary layer separations etc.

The k-ω based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients.

The accuracy of the results depends upon the convergence of the equations. The convergence criteria is set by setting up maximum or RMS value of the residual equations left out which is actually the measure of local imbalance of the conservative control volume equation.

In general, the whole fluid domain is divided into a number of elements, resulting in formation of mesh/grid which may be structured, unstructured or Hybrid depending upon the shape configuration of the geometry. At each connecting point of the element called, node, an equation is generated by using numerical methods such as relaxation techniques/ McCormack techniques etc. For transient state analysis, time marching solutions are used. The element type used for surface mesh is generally, quadrilaterals or triangles and for volume meshing are tetrahedral or hexahedral.

3.3. BODY DESIGN:

The performance of a formula depends highly on the aerodynamic function i.e. its shape which affects the forces being encountered mainly drag and lift forces. The drag force mainly depend s on the frontal area of the vehicle and the lift forces on the overall shape of the body. The main aim in designing a vehicle for aerodynamics is to make it as streamlined as possible to reduce drag being encountered. Since the best shape that has very low drag function is an airfoil, the vehicle is designed as keeping it the basic shape. Now due the varying of streamlines on the body, a lift force is generated, which is not desirable as it reduces the lateral forces, i.e. the forces between the wheels and the road, responsible for the overall performance.

So the next step is to create as much downforce/ negative lift as possible on the front and rear axles. This can be done by using an inverted airfoil shape configuration or using of aids like wings, underbody diffusers or spoilers. Downforce is also important at high speed cornering as it keeps the vehicles on the track.

Since these subsystems are inter-dependent, there is a requirement to carry out the simulation of a complete vehicle in totality to analyze its realistic performance characteristics. Complete simulation also helps in choosing the proper location and size of air inlets, wings, spoilers etc

as well as the body geometry to obtain the best overall performance. A basic simple streamlined body acting as an envelope was first made. However a very important consideration is the interaction between the ground and the vehicle, as the boundary layer formed on the ground also influences the flow at the vehicle. The flow gets separated just after the front bumper which results in creation of undesired additional lift force.

A positive lift can lessen the grip between the road and the tyres. The role of the spoilers is to break the smooth, fast flow above the body, by slowing it down, thus increasing pressure and creating the needed negative lift.

The streamlines would be visualized to check for the formation of wakes and it would be tried to keep the formation of wakes as far away as possible from the body. The formation of wakes at the rear creates low pressure area resulting in suction pressure, thus increasing drag and affecting the vehicle performance.

The drag and downforce creation are both interdependent. Using wings and other aids aimed at increasing downforce also increases drag. So in an efficient design, Drag/Downforce compromise has to be made. The back of the car is also boat-tailed or bent so that flow separation does not take place easily.

Creating the downforce depends upon the body, the front wing assembly and the rear wing assembly.

3.3. 1 CHARACTERISTICS OF FLOW AROUND THE VEHICLE BODY:

Front: stagnation point, overpressure, accelerating flow

Side walls, roof: boundary layer separation depending on the rounding up of leading edges around the front.

Rear wall: in separation bubble nearly constant pressure below the ambient, strong turbulent mixing.

Underbody gap: surrounded by „rough" and moving surfaces, decreasing velocities downstream, sideward outflow.

The vehicles are in the language of aerodynamics known as bluff bodies i.e. they are characterized by boundary layer separation and separation bubbles and drag is mainly caused by pressure forces rather than shear as is in the case of streamlined bodies like airplanes where the flow is attached.

CHAPTER 4

SIMULATIONS

4.1 DESIGN & ANALYSIS OF THE FRAME

The basic frame was designed taking in consideration SAE guidelines and by assembling the frame with the mannequin developed in the previous stage.

The frame has a main roll hoop, front roll hoop, side impact members and other structural members.

The frame was developed on Pro/ Engineer and analyzed for stresses by using COSMOS Express, Solidworks. The structure has been analyzed using the finite element method, where the solid body under consideration in discretized into smaller elements. The size of the elements is controlled to give coarse to fine grid structures. More accurate results are generated if the mesh is made as fine as possible. It is similar to Finite volume method except that a weight function is used. Four impact scenarios were analyzed to ensure the frame design will not fail under steady driving conditions.

- Front Impact
- Rear Impact
- Side Impact
- Roll Over

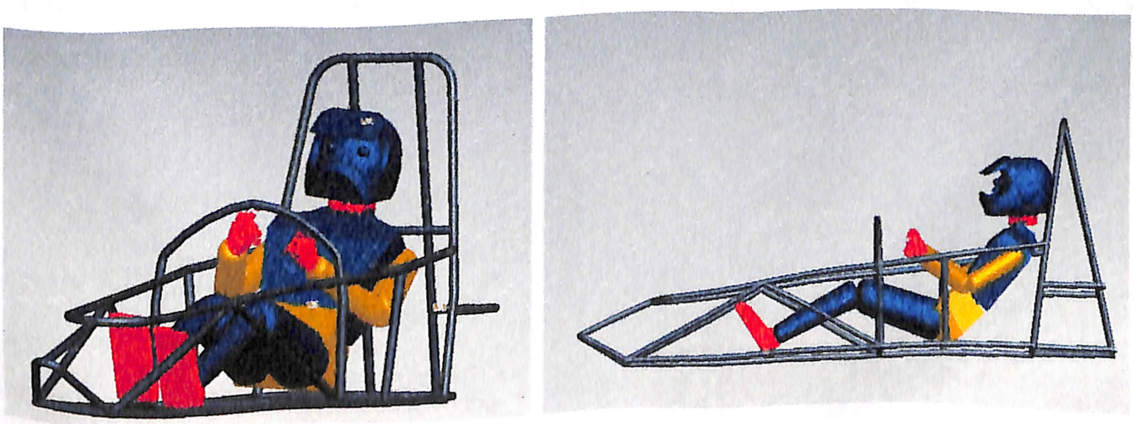


Fig 6: Final frame assembly with the mannequin developed in previous stage.

4.1.2 MATERIAL

The frame material is 4130 N Chromoly Steel with an outer diameter of 1.125" and wall thickness of 0.058" but was modeled as solid rods with1.125" diameter.The material properties are:

Category Tool Steel

Class Carbon steel

Element Weight %

C	0.28-0.33
Mn	0.40-0.60
P	0.035 (max)
S	0.04 (max)
Si	0.15-0.3
Cr	0.8-1.1
Mo	0.15-0.25

Properties Conditions

Density (×1000 kg/m^3) 7.7-8.03

Poisson's Ratio 0.27-0.30

Elastic Modulus (GPa) 190-210

Tensile Strength (Mpa) 560.5

Yield Strength (Mpa)360.6

Elongation (%) 28.2

carburized at 925°C, cooled, reheated to 775°C, water quenched, tempered at 175°C, core properties.

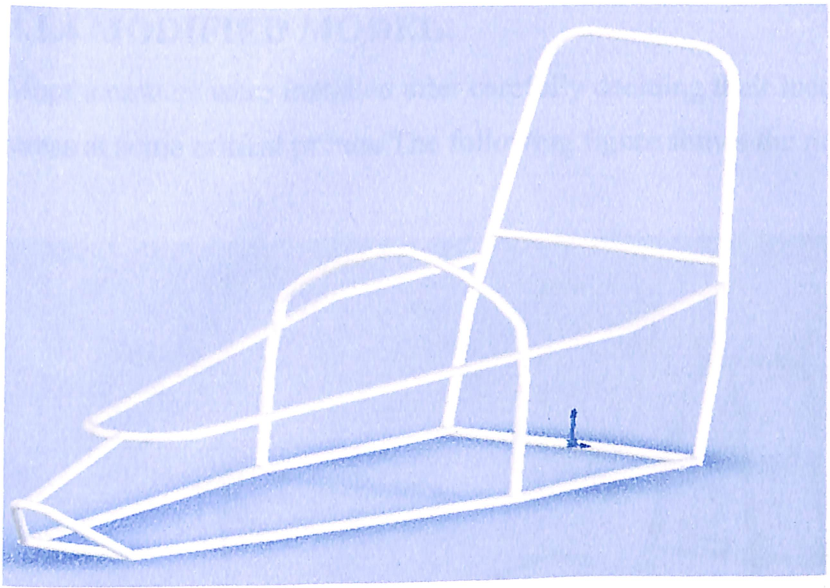


Fig. 7: Basic frame

4.1.3. FRONT IMPACT SCENARIO:

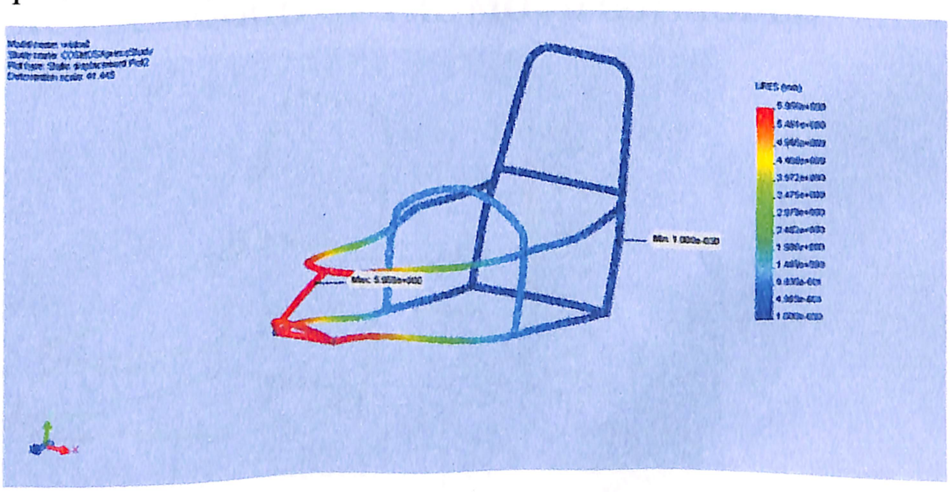
The force was applied on the front most point and the rear being restrained.

ASSUMPTIONS:

Vmax = 100 km/hr (maximum speed)

MTotal = 300kg (mass including driver)

t = 0.1 sec (from full speed to full stop)



v = vi + at

v = final speed

vi = initial velocity

a= deceleration

t= time

a = (v – vi)/t

a = 27m/hr^2

f = MTotal * a = 8000 N

Fig. 8: Displacement model of the basic frame.

4.1.4 MODIFIED MODEL:

More members were installed after carefully deciding their location points to reduce stress at some critical points. The following figure shows the modified frame:

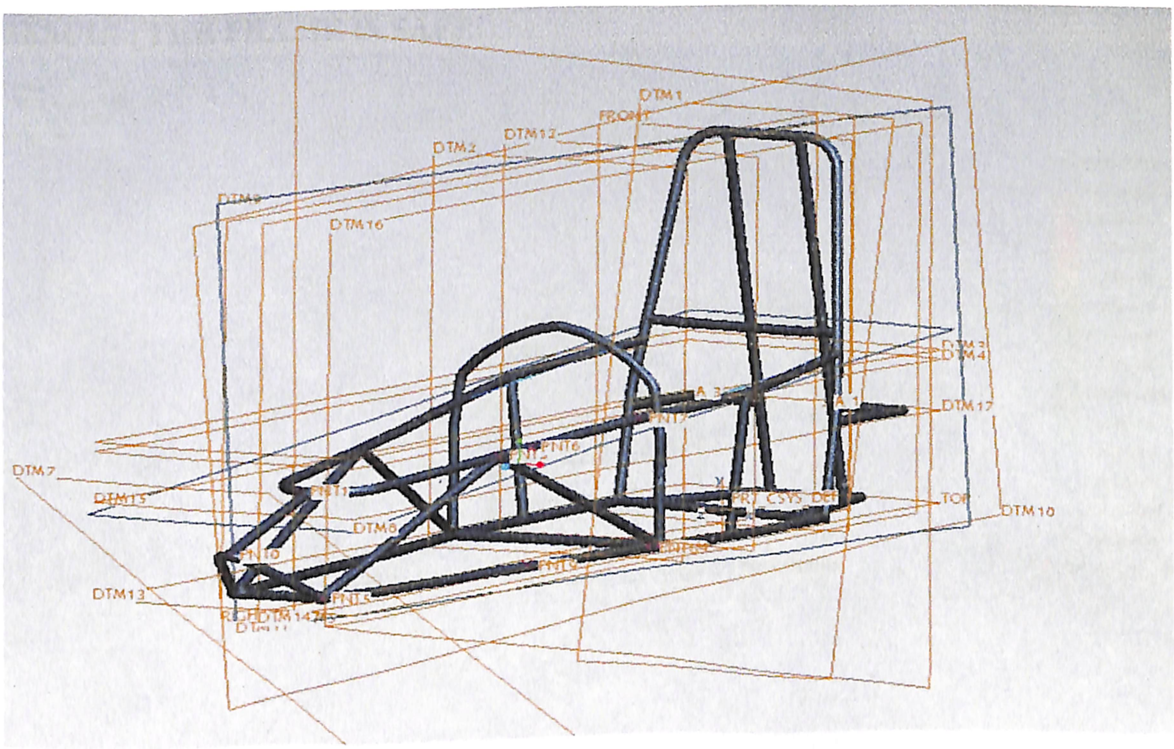


Fig 10: Modified frame

4.1.5 FRONT IMPACT SCENARIO OF THE MODIFIED MODEL

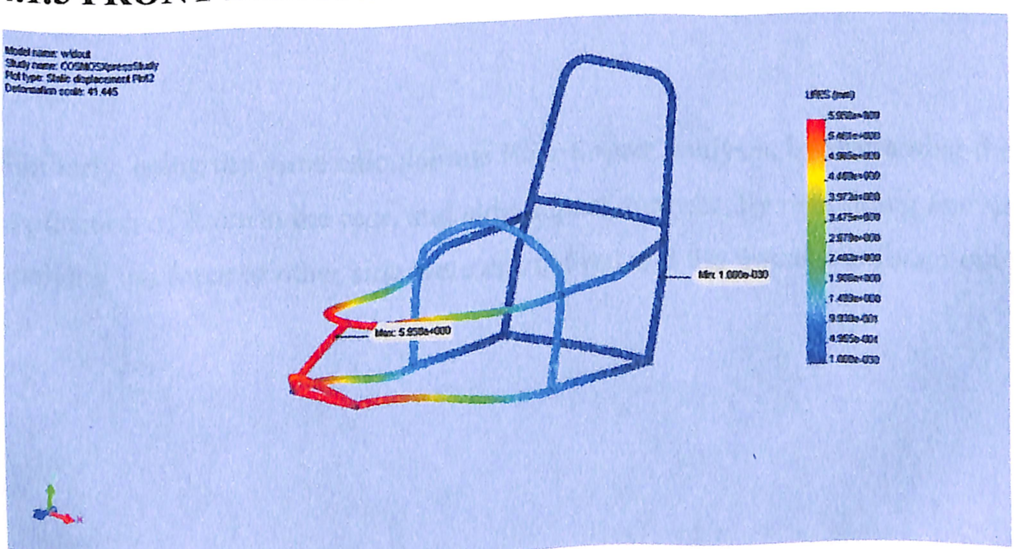


Fig 11: Displacement model of the modified frame

OBSERVATIONS:

MAX STRESS: 6.076e+007 N/m^2

YIELD STRESS: 3.516e+008

RESULT: THE FRAME IS SAFE!

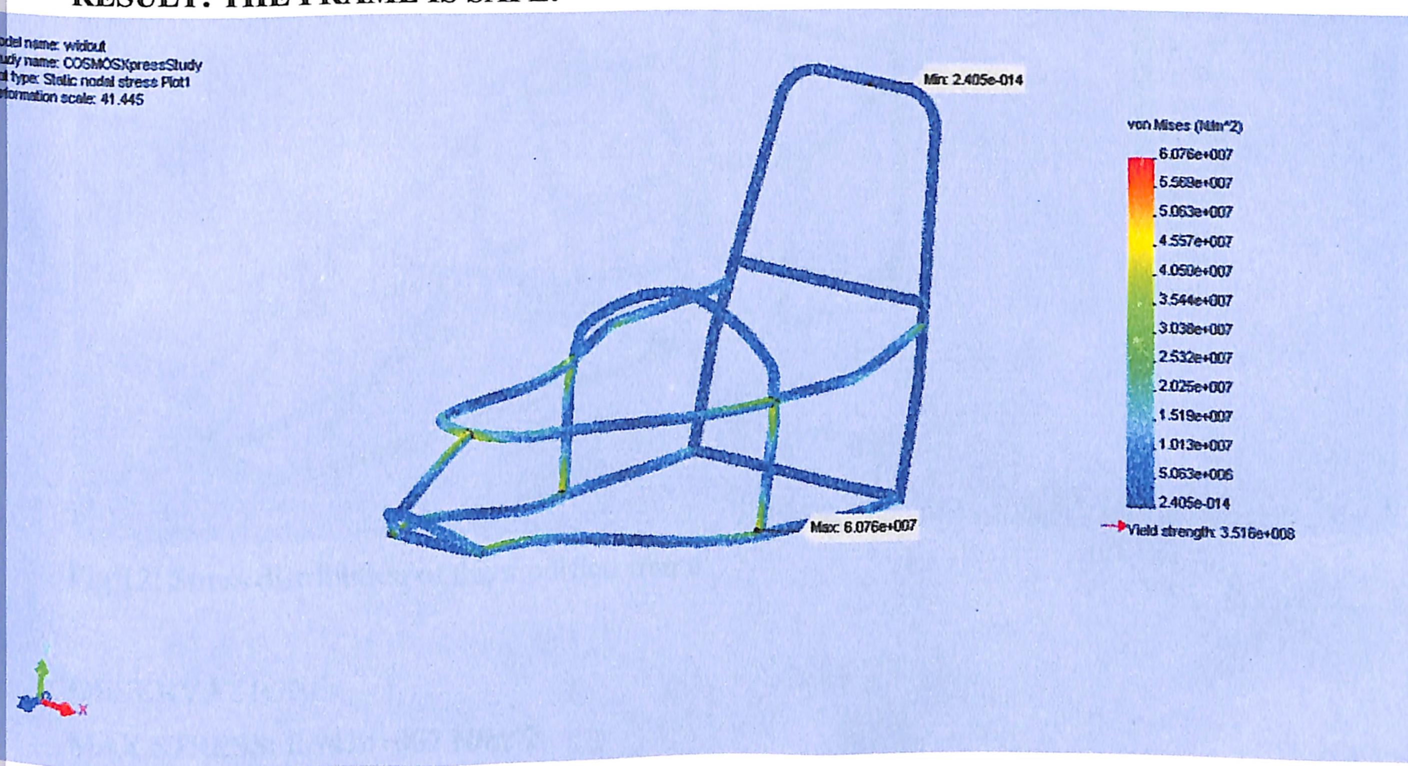


Fig. 9: Stress Distribution on the basic frame

Similarly, using the same calculations Rear impact analysis, by restraining the front and application of force at the rear, and side impact analysis, by restraining one side and applying the force to other side,were carried out and the frame was found out to be safe.

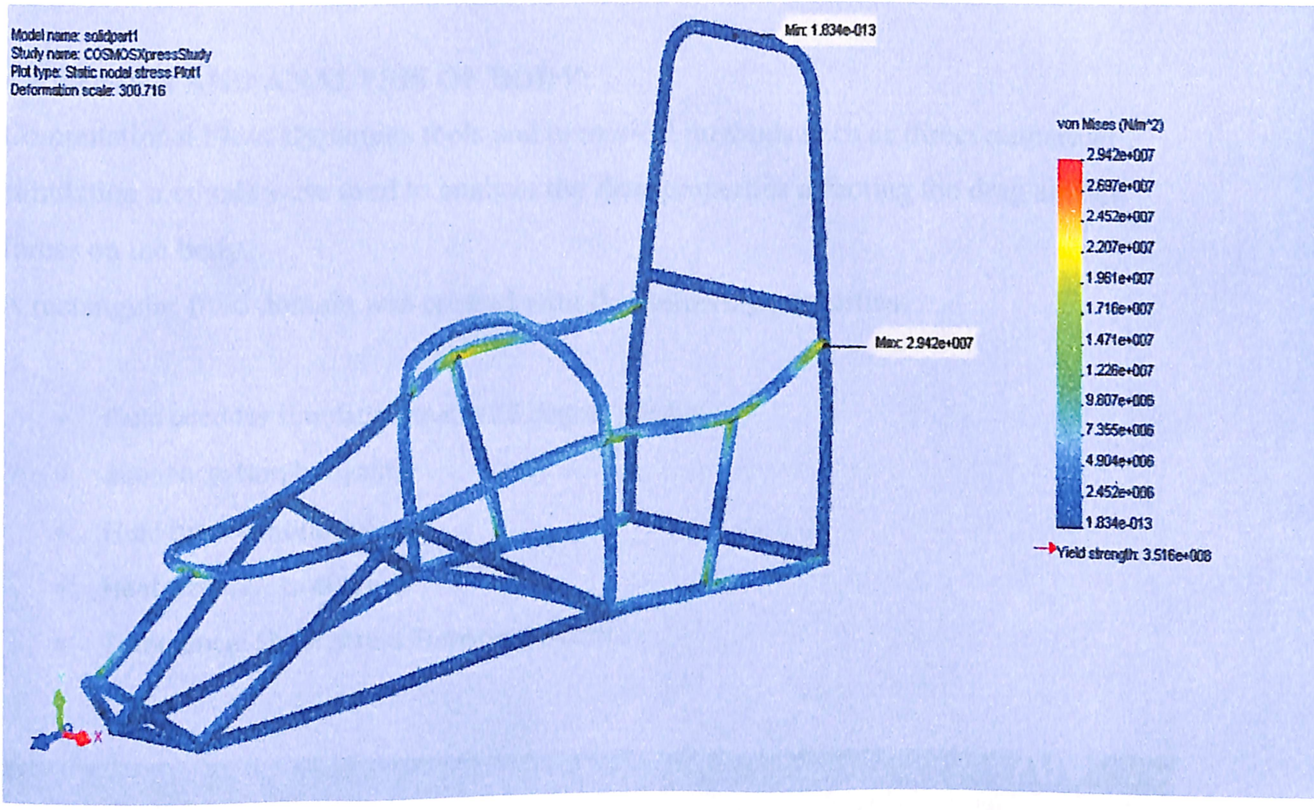


Fig 12: Stress distribution of the modified frame

OBSERVATIONS:

MAX STRESS: 2.942e+007 N/m^2

YIELD STRESS: 3.516e+008

RESULT: THERE HAS BEEN A REDUCTION IN THE STRESS CONCENTRATION AND THE FRAME IS SAFE!

4.2 DESIGN AND ANALYSIS OF BODY:

Computational Fluid Dynamics tools and numerical methods such as direct numerical simulation methods were used to analyze the flow properties affecting the drag and lift forces on the body.

A rectangular fluid domain was created with the following properties:

- Fluid used for simulation :Air at 25 degree Celsius,
- Buoyancy: Non buoyant,
- Fluid type: continuous
- Heat transfer: Isothermal
- Turbulence: Shear Stress Transport model

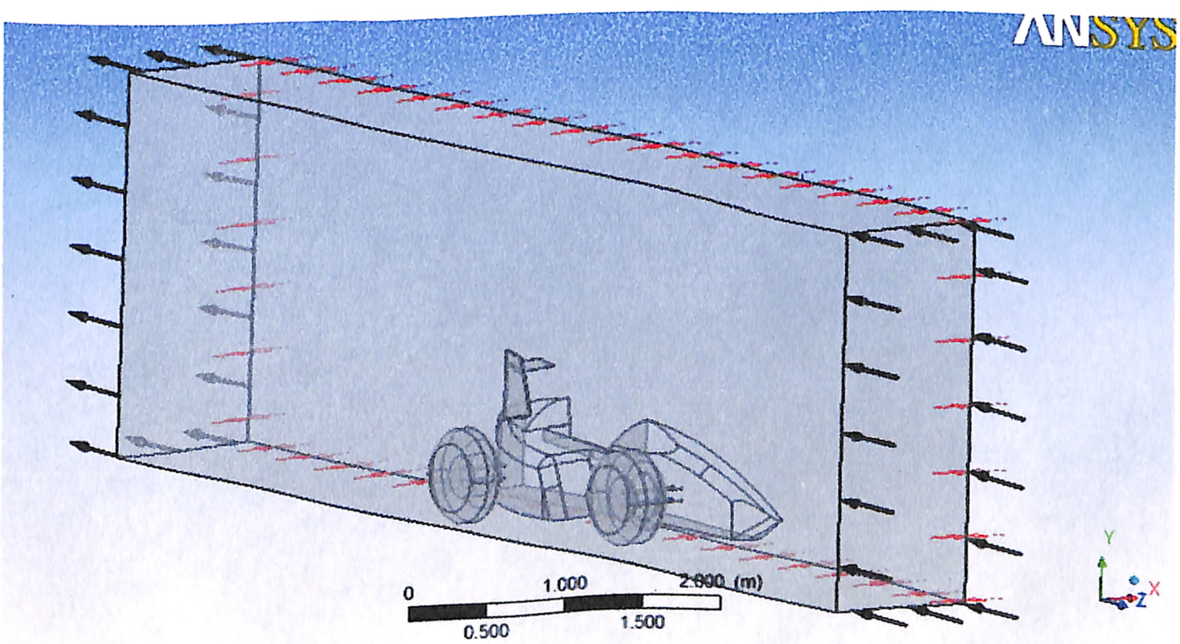


Fig. 13: Rectangular fluid domain with inlet, outlet, body and symmetry regions.

MESHING:

The whole domain is divided into a small number of control volumes, known as elements. Each element is connected to other by connecting points known as nodes and it is here that the solution is generated.

The first step is to generate the surface meshes and then the volumetric mesh. The area of our interest i.e. the body and the fluid flow region is finely meshed by giving mesh controls such as inflation, maximum and minimum element size, radius of influence, growth rate of elements etc. The elements used for surface meshing are Triangles or quadrilaterals and for volumetric meshes are tetrahedral or hexahedral.

Since, our body of interest is a very curvaceous body, triangular elements have been used for surface and we have instilled the use of tetrahedral elements for volume mesh generation as use of hexahedral elements requires very high memory.

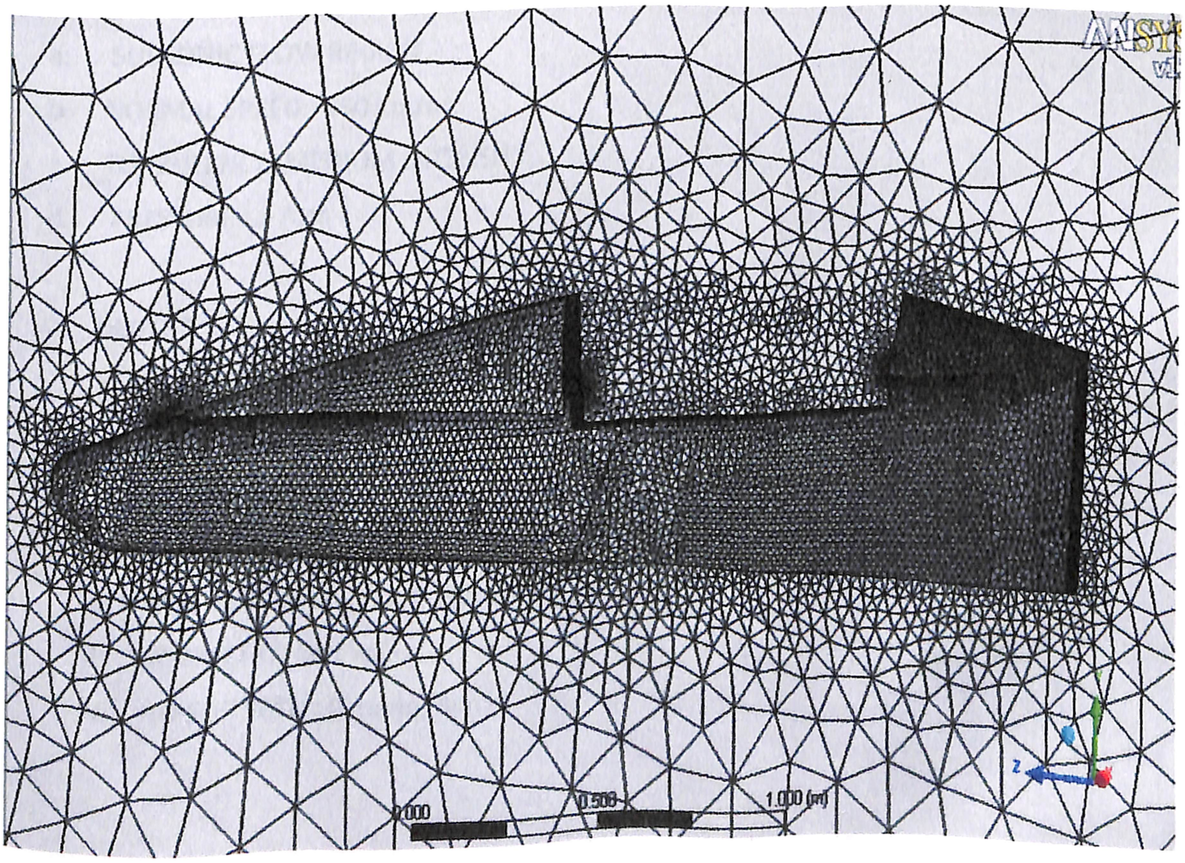


Fig. 14: Volumetric 3d mesh.

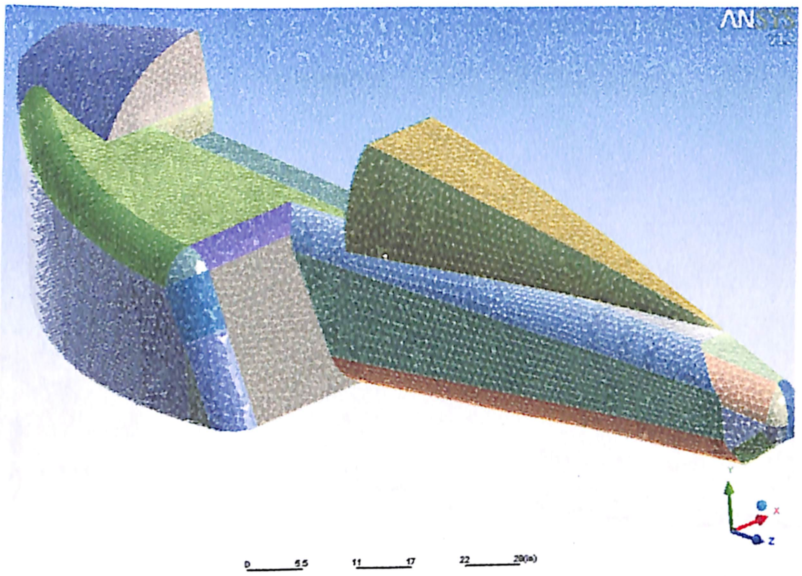


Fig 15: Half surface mesh of the vehicle body.

The Boundary conditions given were:

1. **INLET:**

   a. SUBSONIC FLOW REGIME,

   b. NORMAL SPEED : 150 km/hr,

   c. TURBULENCE: MEDIUM INTENSITY,

   d. PRESSURE : 1 Atm

2. **OUTLET:**

   a. SUBSONIC FLOW REGIME

   b. STATIC PRESSURE: 1 Atm

3. **WALLS:**

   a. FREE SLIP WALLS: Roof, right side

   b. SYMMETRY: Left wall

   c. NO SLIP WALL: Ground, Body

A convergence criteria of residual RMS value 1e-04 was given.

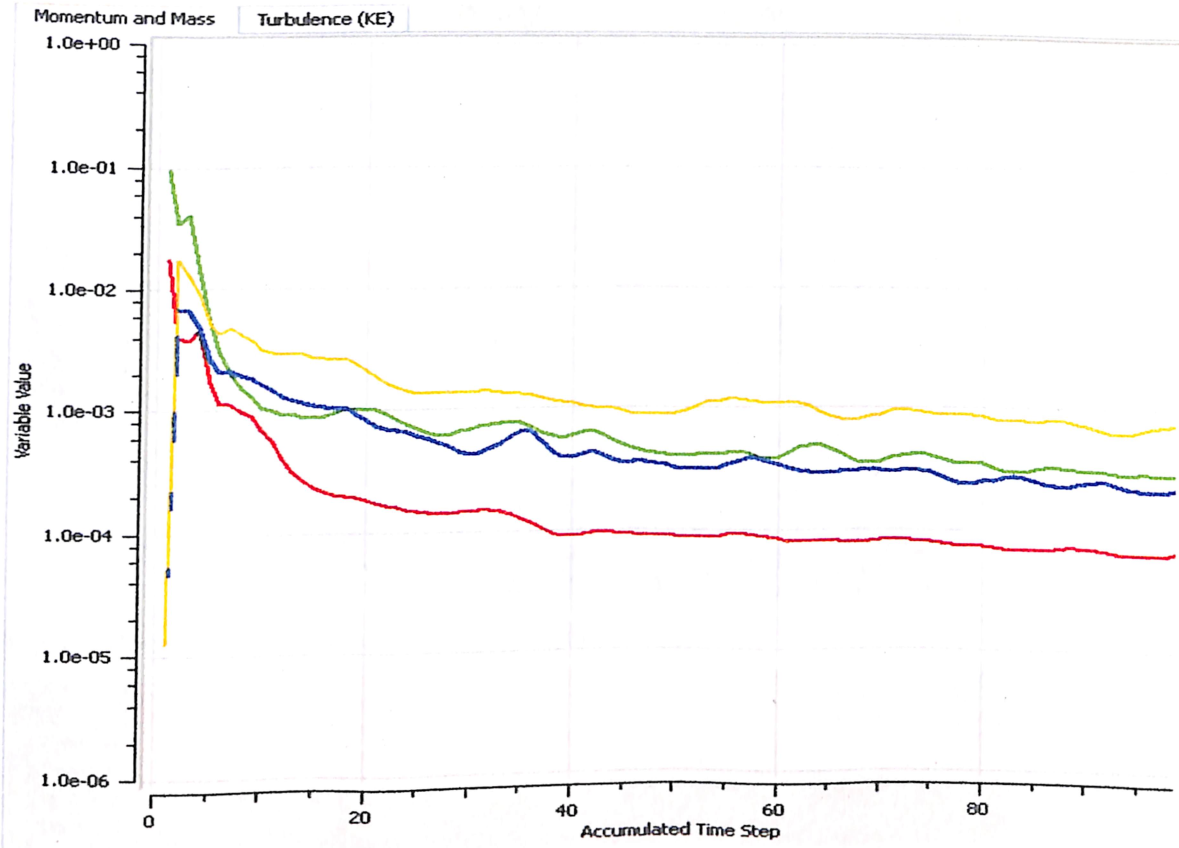


Fig 16: Convergence graph of RMS solution f or momentum and turbulence.

CASE 1: Simple shape, Le Mans type body configuration.(front elevated).

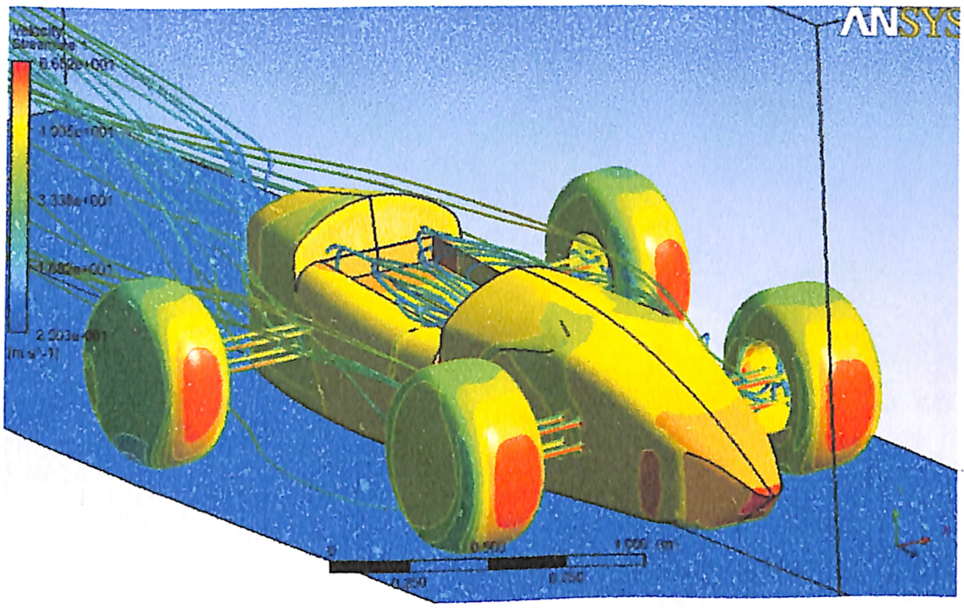


Fig 17: Body pressure contour and streamlines.

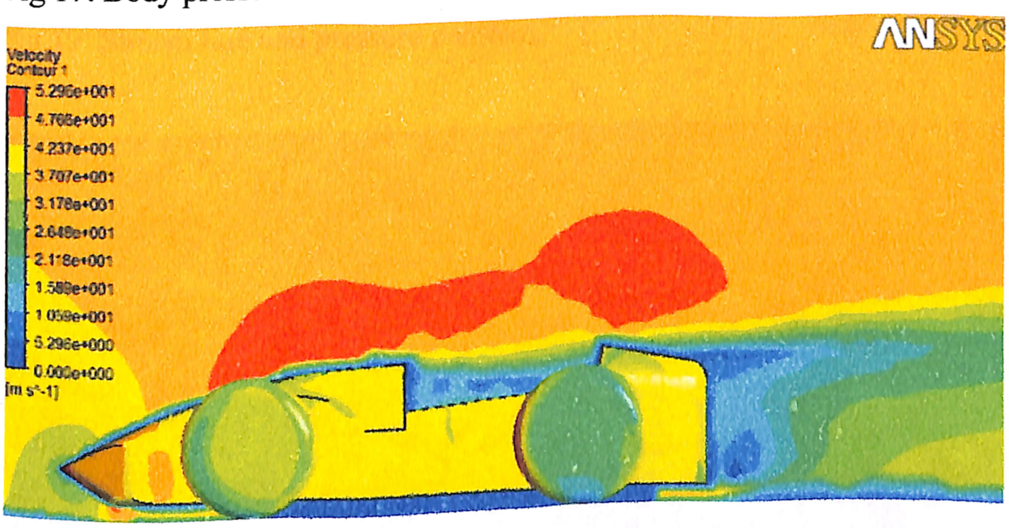


Fig. 18: velocity contour

RESULT:

DRAG FORCE: -326.167 [N]

LIFT FORCE: -39.7843 [N]

OBSERVATION: Wakes can be seen at the rear.

CASE 2: Body with inlets:

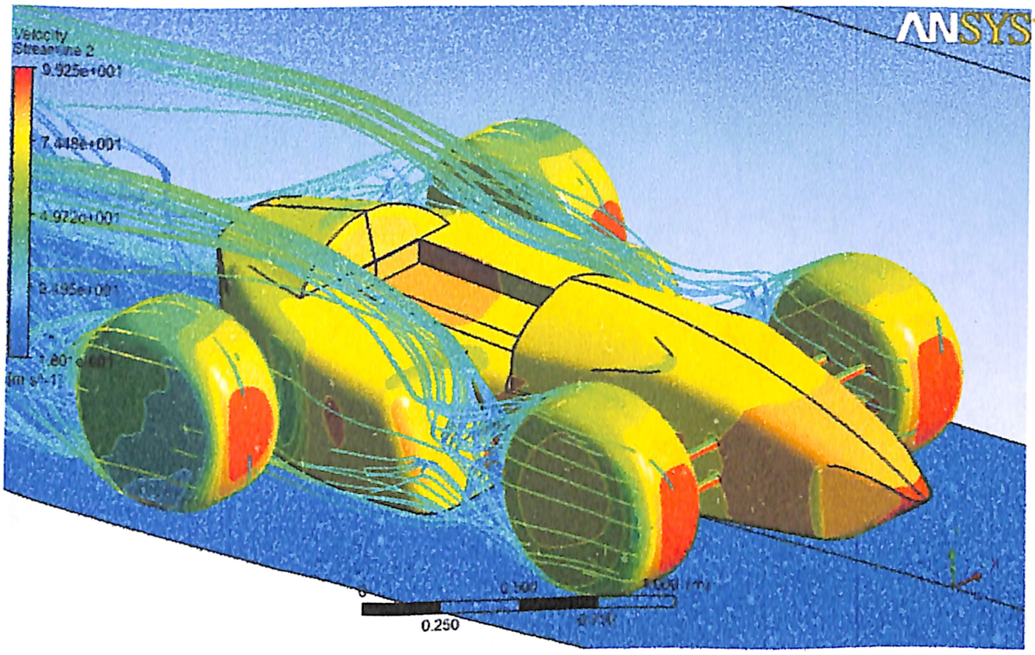


Fig 19: Stream line and pressure contour

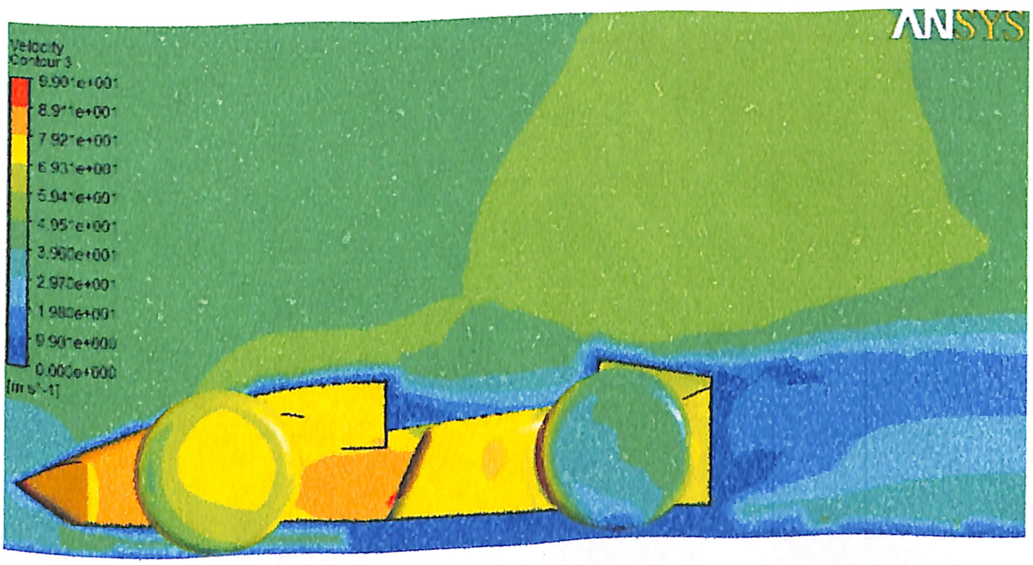


Fig. 20: Velocity contour

RESULT:

DRAG FORCE: -373.086 [N]

LIFT FORCE: - -133.768 [N]

OBSERVATION: With small increment in drag force, a very high increase in downforce has taken place.

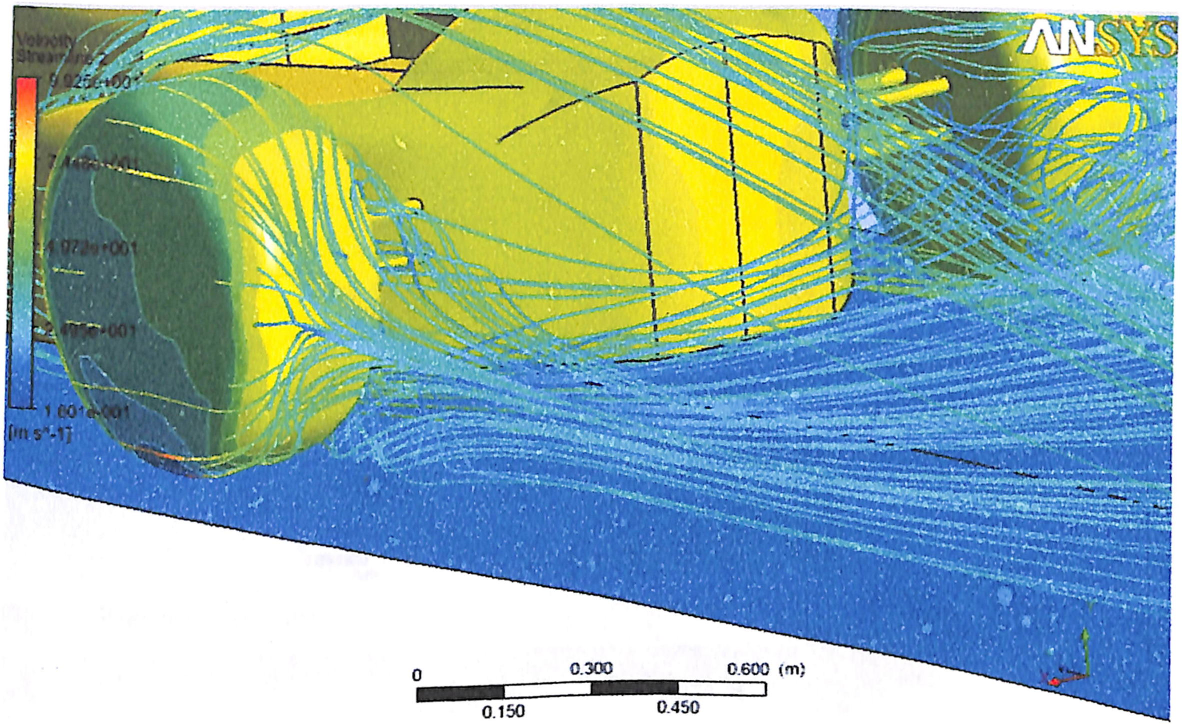


Fig.21: Rotating wheels generating streamlines

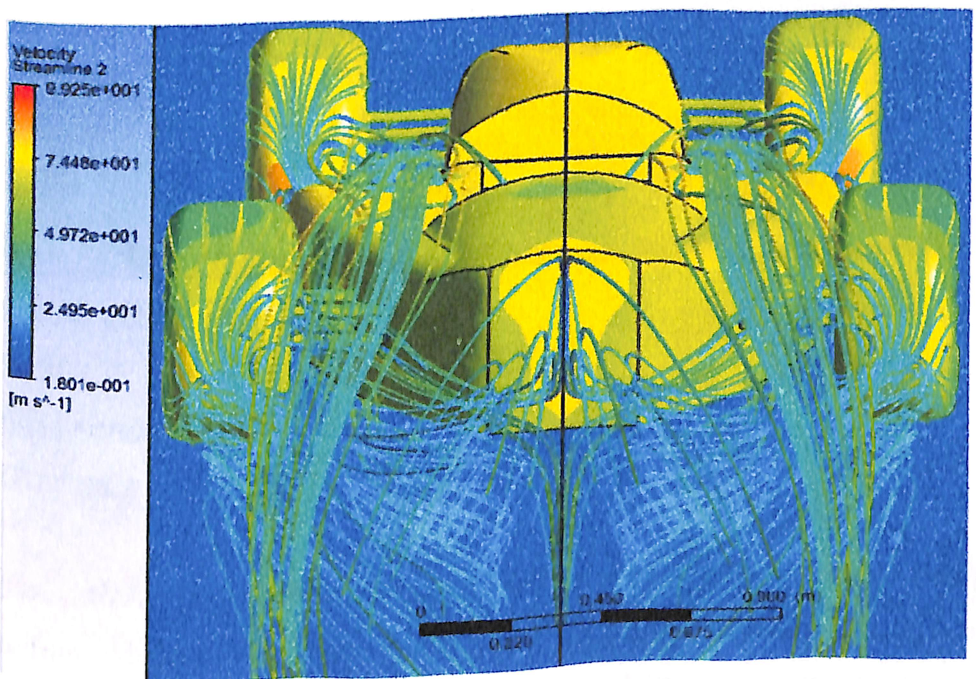


Fig. 22: Vortices being generated at the rear of the body.

OBSERVATION: Effect of rotating wheels on the fluid and formation of vortices at the rear can be seen.

CASE 3: Body with inlet and front down.

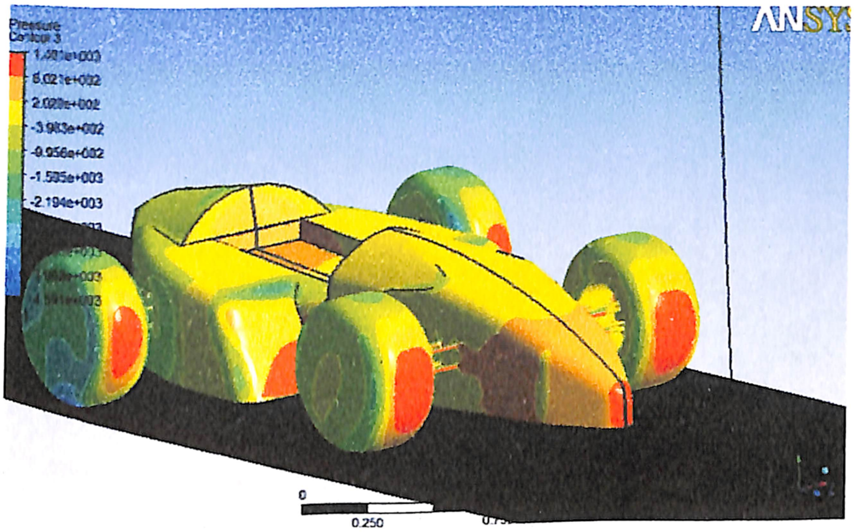


Fig 23: Pressure contour

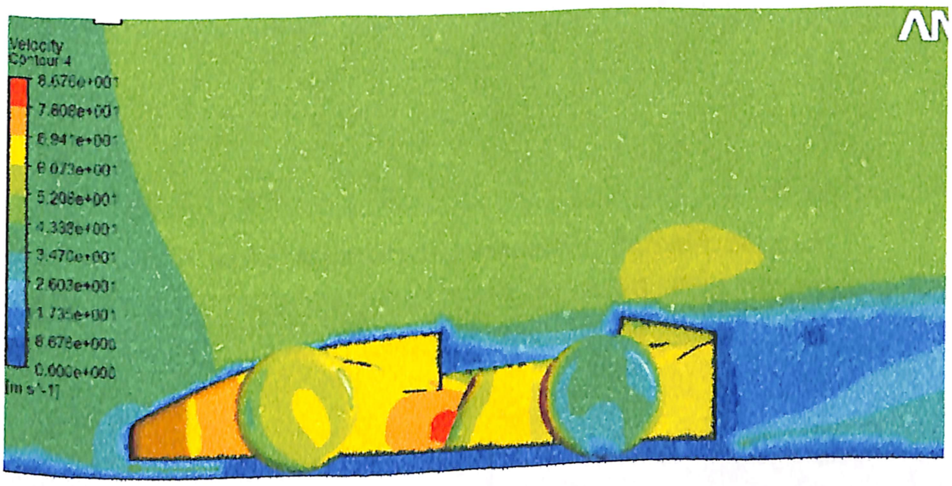


Fig 24: Velocity contour

RESULT:

DRAG FORCE: -347.469 [N]

LIFT FORCE: - -110.273 [N]

OBSERVATION: De-elevating the front has resulted in low downforce as it restricts airflow, whereas the elevated body channelized the flow underneath the body, decreasing pressure, creating suction between road and the body and hence creating downforce.

CASE 4: Body with inlets, rear wings added-inverted NACA airfoil assembly.

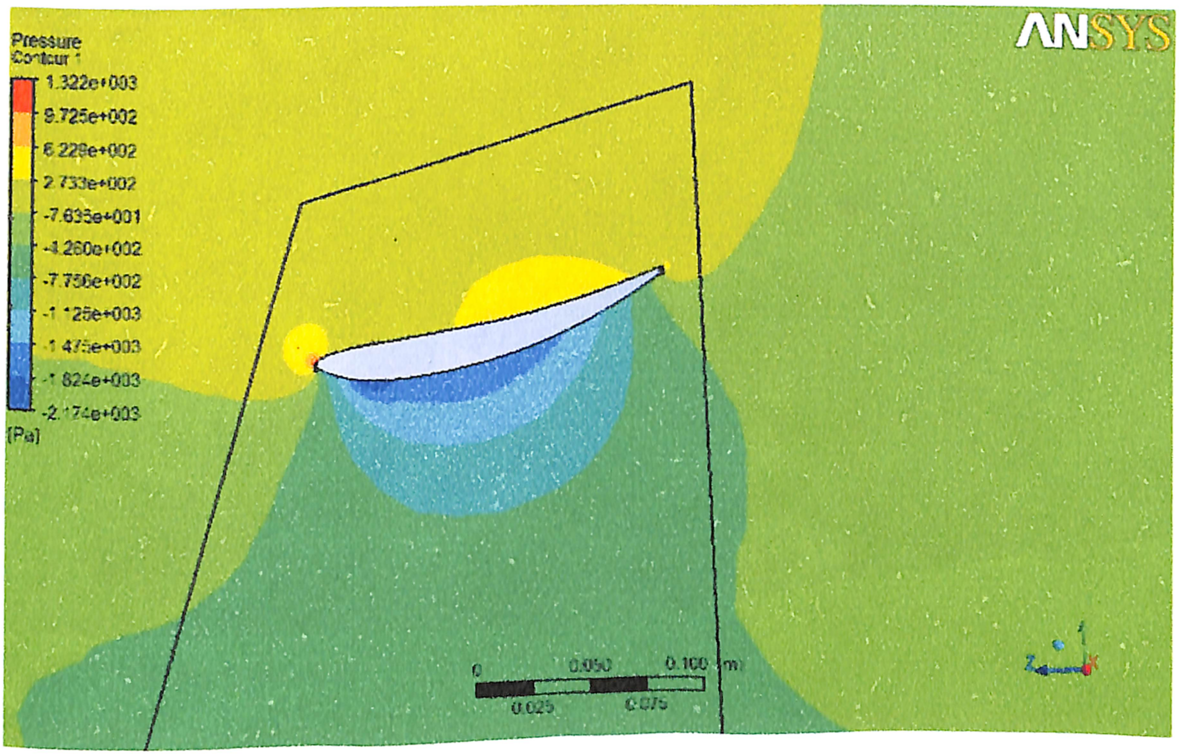


Fig 25: NACA airfoil on rear wing assembly; Pressure contour.

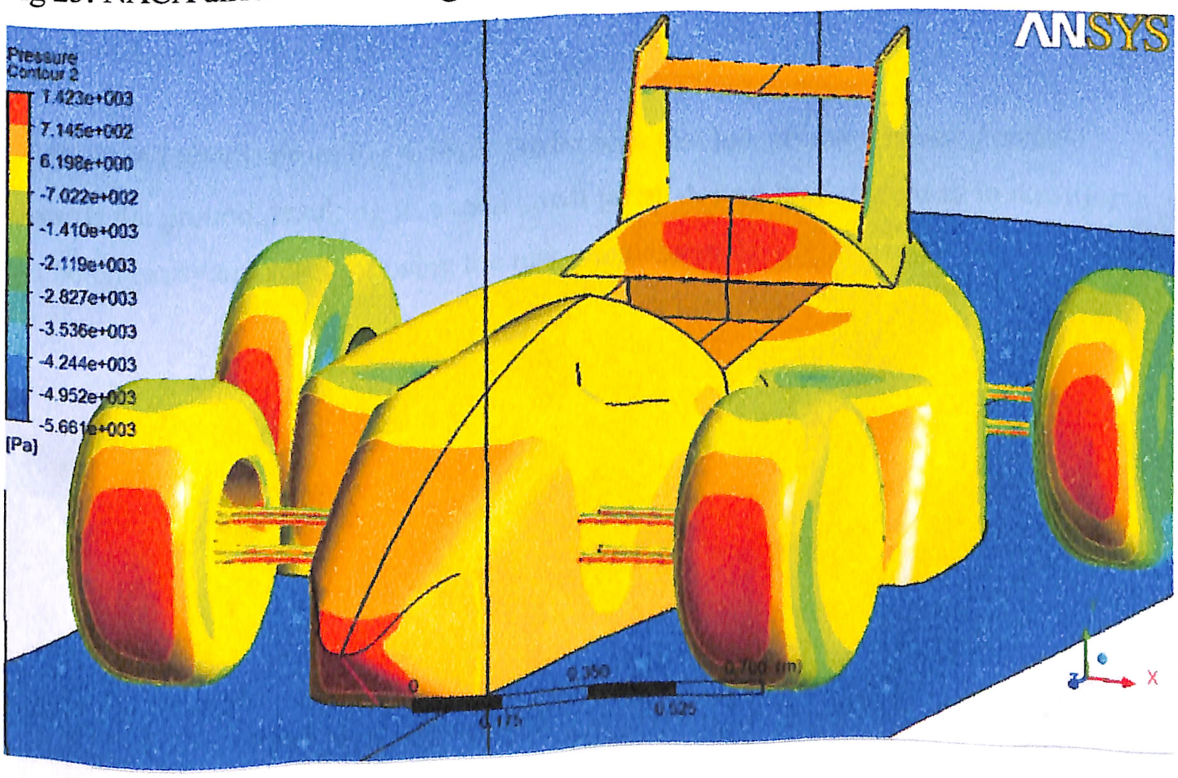


Fig 26: Pressure contour

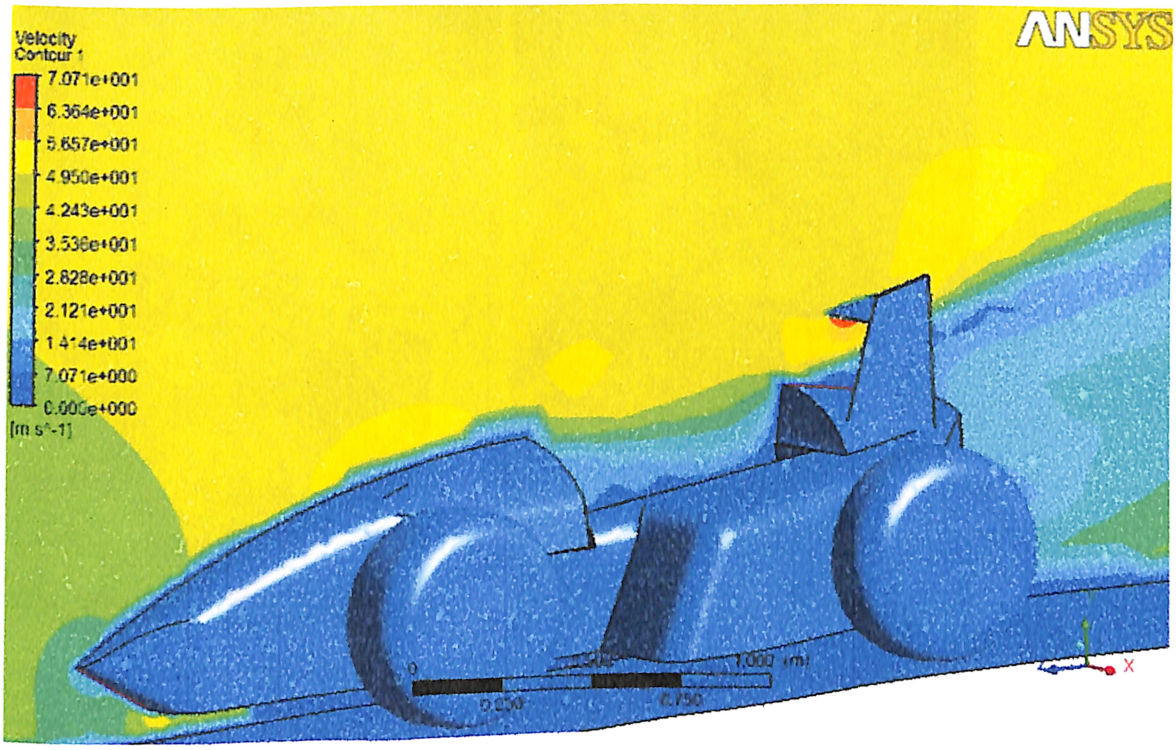


Fig 27: Velocity contour.

RESULT:

DRAG FORCE: -555.913 [N]

LIFT FORCE: - -298.129 [N]

OBSERVATIONS: From Fig 4.18, it can be seen that low pressure is being created towards the ground, resulting in a total down force. A very high increment in downforce has taken place as a result of using the rear wing assembly.

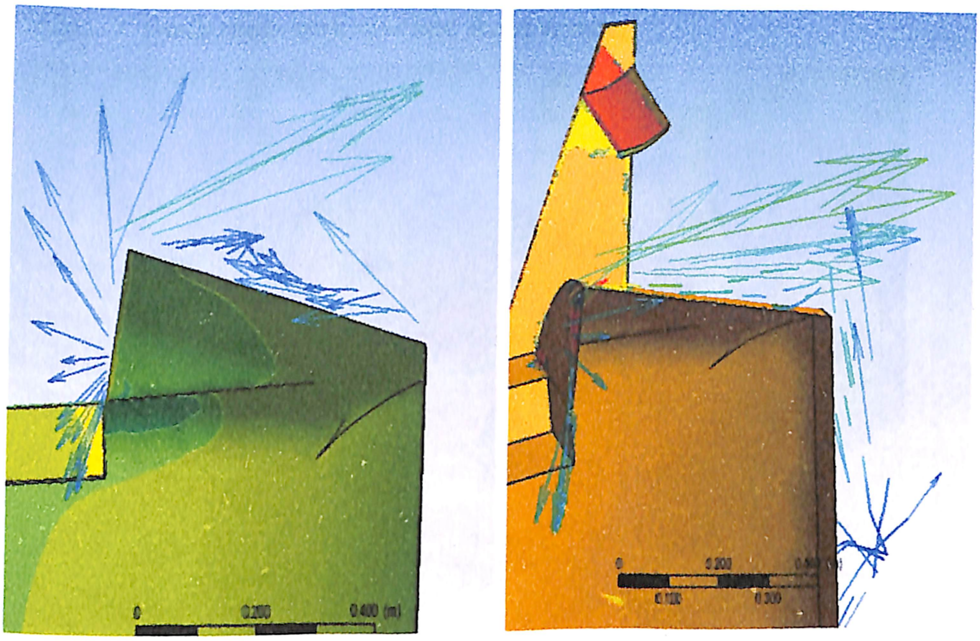


Fig. 28: Velocity contour befor and after installation of wings.

OBSERVATION: The Fig. 4.21 shows us that in the first part, flow separation is taking place on the body itself, it can be seen as the direction of the velocity vectors is changing but due to the installation of the rear wing, no flow separation is taking place and the vectors are not changing their direction.

CASE 5: Body with both rear and front wings.

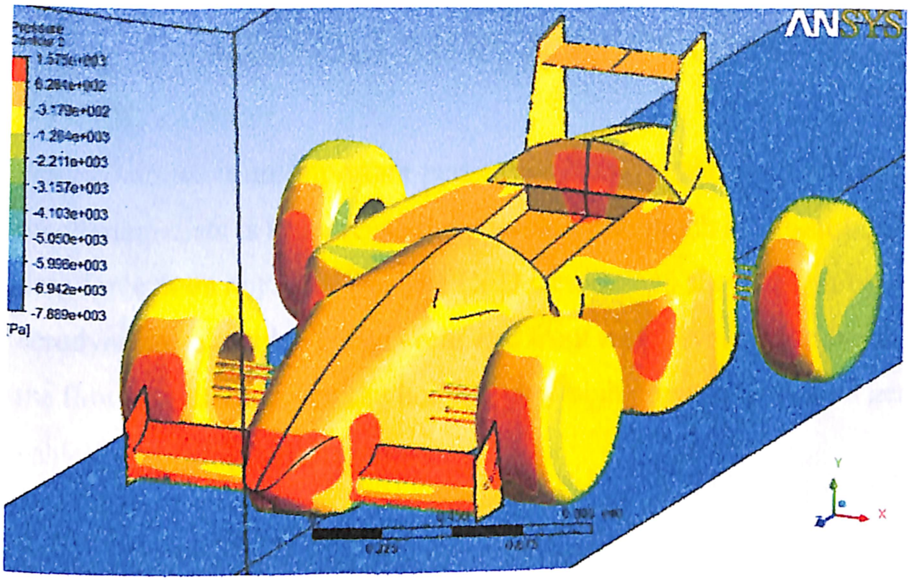


Fig 29: pressure contour on body.

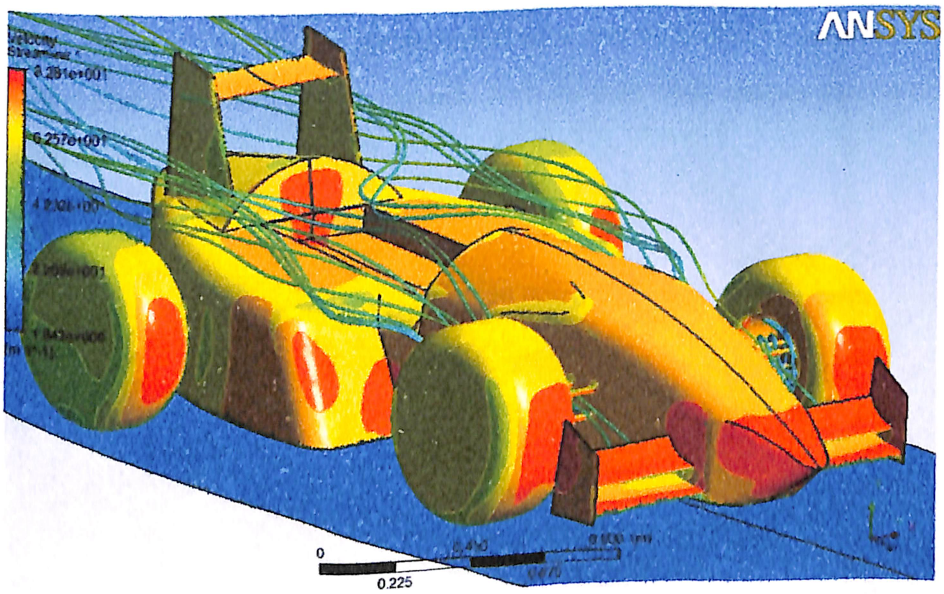


Fig 30: Pressure contour and streamlines.

RESULT:

Drag Force: -608.696 [N]

Lift Force: -382.036 [N]

OBSERVATION: Addition of front wings create additional downforce on the front axle.

CHAPTER 5

CONCLUSION & RECCOMENDATION

5.1 CONCLUSION:

Aerodynamics is an important part of the racing regime and the main aim of all the aerodynamicists is to increase the downforce as much as possible with a condition that drag force does not increase. The CFD simulations showed that by using various aerodynamic aids like Wings- rear and front and elevating the frontal part, so as to make the flow pass underneath the body, a very high downforce can be generated to give the vehicle a very high griping force.

5.2 FUTURE SCOPE OF WORK:

Modifications can be made on the wing assemblies, spoilers, canards etc can be tested for their effects of the flow. Optimization of a vehicle body for aerodynamics aspects is a never ending field and experiments with various body shapes can be done to see their flow field effects.

REFERENCES

1. ANSYS CFX, Release 10.0

2. Ferziger H. Joel, Peric Milovan, "Computational Methods for Fluid Dynamics", Springer, 3rd edition revised.
3. Versteeg H. K., Malalasekara W., "An Introduction to Computational Fluid Dynamics, The Finite Volume Method", Longman Scientific and Technical, Published 1995.

4. Thompson F. J., Warsi Z. U. A. , Mastin Wayne C., "Numerical Grid Generation : Foundations and Applications", Mississippi State, Mississippi, January 1985.

5. S V Patankar, "Numerical Heat Transfer and Fluid Flow", Hemisphere Publication, 1980.

6. *Fundamentals of aerodynamics, SAE.*

7. fundamentals of vehicle dynamics - Thomas D Gillespie
8. Race Car Vehicle Dynamics (Milliken & Milliken)
9. www.wikipedia.org