

Aerodynamic development of a IUPUI Formula SAE specification car with Computational Fluid
Dynamics(CFD) analysis

Ponnappa Bheemaiah Meederira,

Indiana- University Purdue- University. Indianapolis

Aerodynamic development of a IUPUI Formula SAE specification car with Computational Fluid
Dynamics(CFD) analysis

A Directed Project Final Report

Submitted to the Faculty

Of

Purdue School of Engineering and Technology
Indianapolis

By

Ponnappa Bheemaiah Meederira,

In partial fulfillment of the requirements for the
Degree of Master of Science in Technology

Committee Member	Approval Signature	Date
Peter Hylton, Chair Technology	_____	_____
Andrew Borme Technology	_____	_____
Ken Rennels Technology	_____	_____

Table of Contents

1. Abstract	4
2. Introduction.....	5
3. Problem statement.....	7
4. Significance.....	7
5. Literature review	9
6. Purpose.....	13
7. Definitions.....	13
8. Assumptions.....	14
9. Delimitation/scope	15
10. Methodology	15
CFD Set Up.....	15
• Full Car Analysis:	16
• Half car analysis:.....	20
• Wing analysis:.....	23
• Diffuser analysis	26
Radiator set up	28
Aero balance calculation.....	30
11. Results and Findings	32
2015 FSAE design evolution	32
• Summary Table.....	33
• Design Iteration 1:.....	34
• Design iteration 2:.....	38
• Design iteration 3.....	42
• Design iteration 4:.....	51
• Design Iteration 5.....	56
• Design iteration 6.....	59
• Design iteration 7 (Final CFD model)	62
12. Limitations	69
13. Conclusion, Discussion and Recommendation.....	69
14. References:.....	71

Abstract

The research project in question uses the regulations as provide by the Society of Automotive Engineers (SAE) for their yearly racing event Formula SAE (FSAE) competition in Michigan (*2015 FORMULA SAE RULES*. (2014). SAE INTERNATIONAL).The objective is to develop a full aerodynamic package to be produced by the Indiana- university Purdue- University Indianapolis (IUPUI) Formula SAE team in the spring of 2015. The term ‘full aerodynamic package’ consists of the following. A nosecone to cover the front of the car designed to generate ‘downforce’ or reverse lift. An engine cover designed to provide healthy airflow to the rear aerodynamics. A floor or ‘diffuser’ designed to both manage the airflow under the vehicle and also to generate downforce. The diffuser is designed to work with multiple airfoils located at the rear and front of the vehicle.

As wind tunnel testing of these designs is not currently viable, the aerodynamic packages, once designed in Computer aided drawing (CAD), has been evaluated using Computational fluid dynamics (CFD) as a tool for quantifying Lift to Drag ratio, in addition to L/D, the analysis includes code generated to find out the center of pressure (COP) of the car . Multiple iterations of various design concepts have been done to improve/optimize these numbers before a final package is defined. Design tool used to here is Solidworks 2014 and the CFD tool used is STARCCM+ which is a product of CD ADAPCO.

Introduction

The birth of modern automobiles dates back to 1886 when Karl Benz invented a 4 cylinder gasoline engine in Germany. However, since the beginning of 20th century, motor vehicles have undergone a significant improvement in terms of safety, speed and technology (McBeath, 1998). This is due to people's desire to build an advanced vehicle which is called performance cars. However since 20th century, specially build cars were made for racing. Today motorsports has become a huge market and it has become one the most popular sports and attracts a large number of fans. The major Motorsports competition involves NASCAR, Formula 1 racing, Indycar racing and Formula SAE. NASCAR involves modifying the sedan cars for racing whereas Formula1, Indycar and Formula SAE are open wheeled cars competing against each other's. In all these sports cars, the aerodynamic effect has played a huge role in the performance of the vehicle. These days almost all race cars have aerodynamic components in them.

This research project will focus on the Formula SAE intercollegiate competition in which students design, build, test and then race an open wheeled formula style race cars. This competition was introduced in 1981. Since the inception of the competition, the design has been continuously evolving and changing. One development that seems to very common these days is about producing the down force in the FSAE car using various aerodynamic components. Downforce is defined as the downward vertical load that is produced due to aerodynamic load instead of mass of the vehicle (SAE International, SP 1078, 2006). According to Aird(1997),” The tire coefficient of friction increases with increase in downforce, which means that lightweight car will be able to accelerate faster in straight as well as lateral direction”(pg. 107).

Aerodynamic elements produces vertical load on the tires with very little added mass, giving the tires more grip and allowing the car to accelerate faster. The major contributors for aerodynamic downforce over the years are inverted wings and underbody diffuser (Smith, 1985).

The design of the aerodynamic element for race cars is complex due to the body interaction between various parts of the vehicle. Due to the advent of many advanced tools, this complexity has been reduced. Recent advancement in Computational Fluid Dynamics has allowed the simulations of aerodynamics to accurately predict the downforce, flow patterns and other air flow around the vehicles. This simulation tool can greatly reduce the time and cost needed to test aerodynamic elements (Hucho, 2008).

Software's used

Solidworks: The design of each component has been done using Solidworks 2014 software. It is solid modelling CAD software produced by Dassault systems. We have used part design and assembly capabilities of this software.

STARCCM+: CFD is a field of study concerned with the use of high speed digital computers to numerically solve the complete nonlinear partial differential equations governing viscous fluid flow (Freedictionary.com). Computers are used to run the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed computers, better solution is achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing. CFD codes are structured around the numerical algorithm that can tackle fluid flow problem. All codes contain three main elements: 1. a preprocessor, 2. a solver, 3. a post processor.

CD Adapco is a multinational computer software company that authors and distributes applications used for CAD, best known for its CFD products. STARCCM+ is the CFD tool which is a STARCD product.

Problem statement

Aerodynamics plays a very important role in the FSAE car's handling ability and its performance. IUPUI FSAE team is implementing full aero package for the competition in Michigan 2015. This is the first time our team is going with aero package. In order to implement these components, an in depth analysis has to be done using the tools available at our university. Due to limited budget, team does not have access to wind tunnel. Therefore, the complete analysis has been done using CFD as a developing tool. This project is focused on developing and conducting a thorough analysis of the aerodynamic component implemented using CFD. This project also explains the formulation of center of pressure which is a critical aspect to determine the handling ability of the car.

Significance

According to McBeath(1998), "There is perhaps, no other aspect of competition car technology that has had as big an influence on performance as exploitation of downforce"(p. 25). Adding the aerodynamic components to the car can result in reducing drag and increasing downforce. Adding aerodynamic components improves both the performance as well as handling (Katz,

1994). Reduction in drag yields a better fuel economy and also the car acceleration is improved. Increase in the downforce results in better utilization of tire grip. This results in better handling of the car at higher speeds. It also improves the turning ability of cars at high speed due to more usage of tyre grip. Thus aerodynamics plays a huge role in how a car performs in a race.

According to Smith(1984),”Downforce describes the downward pressure created by the aerodynamic characteristics of a car that allow it to travel faster through a corner by holding the car to the track or road surface”(pg 234). Some elements to increase vehicle downforce will also increase drag. It is very important to produce a good downward aerodynamic force because it affects the car speed and traction (Aird, 1997). In addition to provide increased adhesion, car aerodynamics are frequently designed to compensate for the inherent increase in oversteer as cornering speed increases. When a car corners, it must rotate about its vertical axis as well as translates its center of mass in an arc (Smith, 1984). However in a tight radius corner the angular velocity of the car is so high, while in longer radius corner the angular velocity is much lower. Thus, the front tires have more difficult time overcoming the car’s moment of inertia during corner entry at low speed and much less difficulty as the cornering speed increases. So the natural tendency of any car is to understeer at on entry to low speed corners and to oversteer on entry to high speed corners. To compensate for this unavoidable effect, car designer often bias the car handling toward less corner entry understeer and add rearward bias to the aerodynamic downforce to compensate in higher speed corners. The rearward aerodynamic bias can be achieved by using a diffuser and rear wings (Wong, 1993). For an FSAE car, where there are too many turns and corners, the vehicle handling becomes extremely important to win the competition. A good Formula SAE vehicle is judged by various parameters and aerodynamics plays a huge role in providing appropriate performance.

Literature review

In the past years the aerodynamics aspect of the FSAE car was neglected due to its slow speed operation. In recent past, the teams have realized the importance of Aerodynamics on the FSAE cars. Teams have understood the value of aerodynamic influence on the car's handling and cornering abilities.

Various research papers have been published on the use of CFD for the design of SAE aerodynamics. Underbody diffuser has been the area of major research as it forms a major source of downforce in the car with minimal drag. It has a very high aerodynamic efficiency (Downforce/Drag). Sometimes at a ratio of 200:1 to 400:1 (McBeath, 1998). The idea behind an underbody diffuser is to use the close proximity of the vehicle to the ground, termed ground effect, to cause a venturi-like effect under the vehicle (Wong, 1993). Like a venturi, there is a nozzle that increases the velocity of air underneath the vehicle, a throat where the maximum velocity is reached and a diffuser where the air is slowed back down to the free stream velocity. Bernoulli's equation shows us that as the velocity increases relative to the free stream velocity the local pressure is decreased (Aird, 1997). Using this lower pressure under the vehicle and higher pressure on the top, downforce can be created.

Like the venturi, the efficiency of an underbody is only as good as the efficiency of the diffuser section (Smith, 1984). Due to its high visibility relative to the rest of the underbody, there is little common misconception in the race car industry to how diffuser works. First is that diffuser is what actually creates all the downforce and second is that the diffuser expands the air under the vehicle causing lowered pressure. Both of these concept are false since the role of the diffuser is to slow the air under the vehicle back down to free stream to reduce the drag and

increase the overall underbody efficiency, and as it is an open system with gaps around the edges it is unable to expand the air to cause a density change (Aird, 1997). With these things in mind, it is the diffuser angle and the entrance location that drives the underbody performance. The location of the entrance of the diffuser greatly affects where the low pressure occurs on the vehicle underbody (McBeath 2006). The center of pressure can be moved forward or rearward. For a race car the balance is critical to the vehicle as it determines the understeer and oversteer characteristics.

In general it is desired to have the highest angle without flow separation to generate maximum downforce. Once separation occurs the downforce is reduced and drag is greatly increased (Wong, 1993). Two dimensional simulation of the diffuser angle shows maximum downforce is reached with an angle of 7 degree (McBeath 2006).

However, in experiment and 3d simulation there is another effect that is occurring that changes this. Starting at the diffuser entrance there is a vortex that forms that travels down the length of the diffuser. A vortex adds 4 rotational components to the velocity decreasing the pressure along its length (SP 1078, 1995). This vortex flow also adds energy to the flow and will delay the separation allowing larger diffuser angles. This effect has been utilized in this our designing of underbody diffuser. Vortices can also be used on other parts of the underbody. Large vortex generators can be placed at the entrance of the underbody so that the vortices travel along the length of the vehicle, reducing the pressure and increasing downforce. These vortices can also be used along the sides of the underbody creating a false seal that also increases downforce (Smith, 1984). All of these ideas can be used together to create an effective underbody that will produce large amounts of downforce with a relative small increase in drag (Aird, 1997). The problem that occurs however is that there is complex interaction between all

parts of the underbody as well as the car body, making design an uncertain area. Also, since racing is a competitive sport, most of the specific information about under tray design is not published.

In our initial iteration we used the NACA 4 digit airfoil for shaping our wing section.

The **NACA** airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of the NACA airfoils is described using a series of digits following the word "NACA". The parameters in the numerical code can be entered into equations to precisely generate the cross-section of the airfoil and calculate its properties. (Jacob, Ward, Pickerton (1933)). The details of the wing section can be found on the internet. Below Diagram shows the NACA profile and how they are named. Most of the teams utilize the NACA airfoil library to choose the optimum shape for their car.

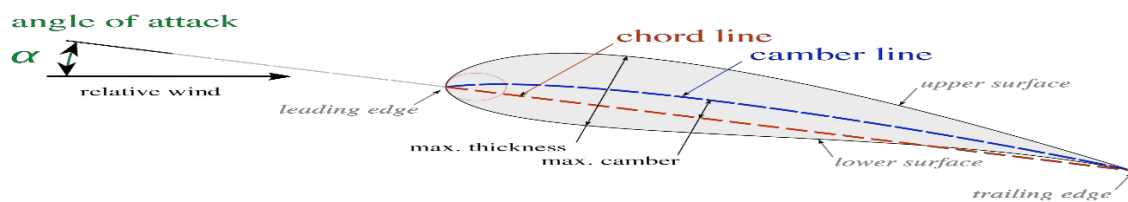


Figure 1: Wing profile

The shape of the NACA airfoils is described using a series of digits following the word "NACA". The parameters in the numerical code can be entered into equations to precisely generate the cross section of the airfoil and calculate its properties (www.airfoiltools.com).

Figure 1 shows the profile of a wing. NACA four digit wings are also similar to the profile shown in the above figure. NACA four digit wing sections define the profile by:

- First digit (M) describing maximum camber as percentage of the chord.
- Second digit (P) describes the distance of maximum camber from the airfoil leading edge in tens of percent of the chord.
- Last two digits (XX) describing maximum thickness of the airfoil as percent of the chord

There are few SAE paper published which describes the usage of CFD on the development of FSAE car. Not much literature is found on how to calculate the center of pressure, usage of the CFD result to further improve the design. This paper addresses both the above issues as well.

The front end device used here are front wings and gurney flap. The Gurney Flap is a small tab projecting from the trailing edge of a wing which is typically placed at a right angle to the high pressure side surface of the airfoil, and generally is $\frac{1}{2}$ " or $\frac{3}{4}$ " on race car wings (McBeath, 1998). This trailing edge device can improve the performance of a simple airfoil to nearly the same level as a complex high-performance design. The basic objectives of these devices are to minimize the disturbance or wake which is headed upstream, Enhance/maximize cooling intake/exit and provide balance adjustment.

The rear end device used on this car is rear wings, end plates and rear Gurney Flap. The functions of these devices is to generate rear load, minimize rear bodywork lift and rear wheel wake impact, Enhance cooling intake/exit and improve the underbody flow (McBeath, 2006). The bodywork includes nosecone and engine cover whose primary function is to generate some downforce, minimize lift, house cooling radiator, provide air intake to the engine and house engine/gearbox/driver etc.

Purpose

This project serves various purposes. First and foremost, it is very important for us to understand the effect of aerodynamic components on the Formula SAE car. Once we understand its effects, team can then come with an optimized aero package for the competition. This in-depth process will separate our car from others in the competition. The team will have necessary and sufficient data to prove the aerodynamic design implemented on the car is worthwhile and provides a large advantage to the car's performance. A full analysis of the aerodynamic package has been done which includes theoretical research, CFD simulations and the data analysis.

Our fully analyzed aerodynamic package-accompanied by CFD simulation results and other important data will support our "design presentation" aspect of the competition, enabling us to score higher in the static part of the competition. The increased downforce and a better balanced car by the aero package will help our car perform better in the dynamic part of the events, giving us an edge the team never had before.

Definitions

1. **Drag:** drag refers to forces acting opposite to the relative motion of any object moving with respect to a surrounding fluid (www.gmecca.com).
2. **Downforce:** It's a downwards thrust created by the aerodynamic characteristics of a car. The purpose of downforce is to allow a car to travel faster through a corner by increasing (www.gmecca.com).
3. **Center of pressure:** Center of pressure is a point about which moment due to aerodynamic forces in all direction is zero. The distance between the COP and

Center of gravity of the car determines the stability of the car. Closer the distance better is the stability.

4. **Pressure recovery:** It's a term which will be used for the diffuser. It is referred to how well the diffuser recovers from the low air pressure at the throat exit (diffuser inlet) to the ambient pressure at the diffuser exit (Clancy, 1975).
5. **Computational domain (CFD):** It's the geometrical region over which a simulation is performed. (www.nafems.org)
6. **Mesh:** A mesh is a discretized representation of the computational domain, which the physics solvers use to provide a numerical solution. There are different meshing strategies, and each one has its pros and cons, being more suited for one or other application.(www.Nafems.org)
7. **Vortex:** In fluid dynamics, a **vortex** is a region, in a fluid medium, in which the flow is mostly rotating on an axis line, the **vortical flow** that occurs either on a straight-axis or a curved-axis. (Ting,1991)
8. **Incompressible flow:** It is also called as isochoric flow. It refers to the flow in which the density of the fluid is constant (Hucho,1998)

Assumptions

- CFD results are assumed to be in ideal environmental conditions. We are neglecting the effects of outside environment when the car is on track.
- The air flow is incompressible and non-viscous.
- There is no energy loss due to friction between the air and the car body surface.
- There is no heat energy transferred across the boundaries of the car to the air.

- The airflow is in steady state.
- CFD simulation results are assumed as actual working conditions.

Delimitation/scope

- All the data in this research paper are simulation results. No correlation work has been done to real time data. Thus, we assume that the simulation data are correct considering the recent advancement in CFD world.
- The CFD simulation only include straight line condition. No yawing condition is simulated
- Due to the limited budget, our team does not have access to wind tunnel. Thus, there is no comparison data for the CFD result.
- CFD software only simulates the condition of the road; it does not reproduce them exactly.(Hucho,1998)
- The temperature stratification above the road is not reproduced in the simulation(Hucho,1998)
- Oncoming flow to the car is uniform therefore the turbulence level is low (Hucho,1998)

Methodology

CFD Set Up

As mentioned earlier, a CFD code works on three elements: Preprocessor; Solver and Postprocessor. This section explains the different types of CFD code set up.

4 different types of set up were done for this project:

1. Full car analysis including radiator and exhaust muffler
2. Half car analysis excluding the exhaust muffler and radiator
3. Wing analysis
4. Diffuser analysis

Full Car Analysis:

If the car is non symmetric about its center, then we have to run the CFD analysis on the whole car.

Figure 2 below shows the CAD model of the unsymmetrical car body comprising of radiator duct and exhaust muffler. The below figure shows the unsymmetrical model which consist of exhaust muffler and radiator duct.

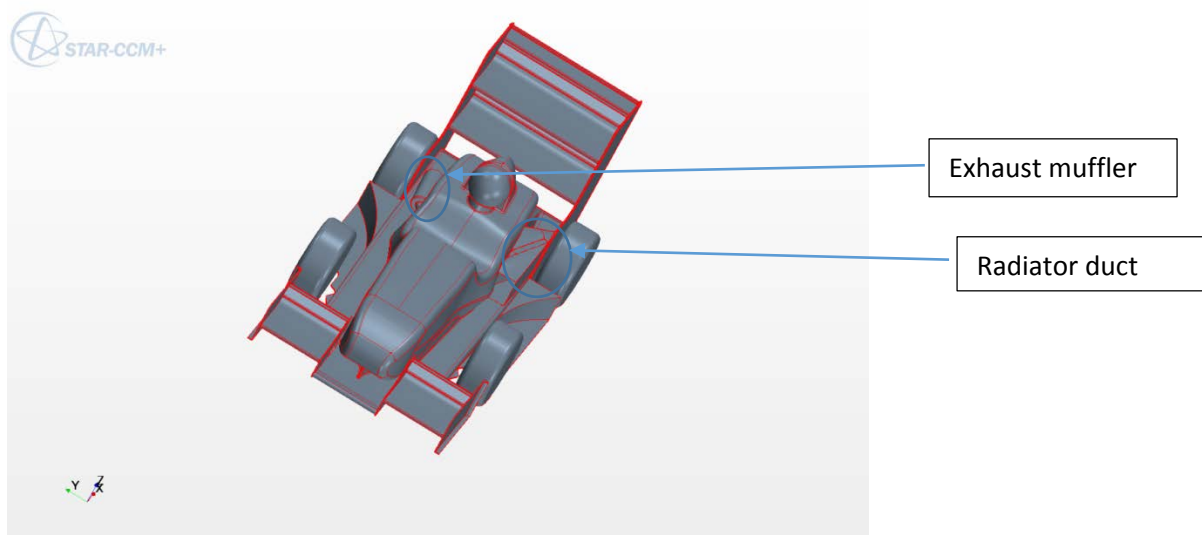


Figure 2: unsymmetrical car body

The table below shows the preprocessor, solver and post processor set up.

Preprocessor	Solver	Post processor
<ul style="list-style-type: none"> • Velocity Inlet: 30mph • Pressure outlet 	<ul style="list-style-type: none"> • 3 dimensional flow 	<ul style="list-style-type: none"> • Downforce • Drag

<ul style="list-style-type: none"> • Floor: moving wall condition at 30 mph • Far walls 		<ul style="list-style-type: none"> • Drag and downforce coefficient • Front balance
<ul style="list-style-type: none"> • Wheel set rotating at 460 rpm 	<ul style="list-style-type: none"> • K-omega RANS model 	<ul style="list-style-type: none"> • Scalar plots • Vector plots
<ul style="list-style-type: none"> • Mesh type: Hexahedral (trimmer) • Surface mesh size: 10 mm • Volume mesh size: 1000mm • Volume growth rate: Very slow • Surface growth rate: Very slow • Volumetric control 1: 50 mm • Volumetric control(cone): 100mm 	<ul style="list-style-type: none"> • Segregated flow • Steady state 	<ul style="list-style-type: none"> • Streamlines • Resampled volume • Threshold

Meshing is the most important part of a CFD set up in order to capture the flow details. Finer the mesh, the accuracy of the result is better. The downside of fine mesh is the increase in computational time. Therefore, we had to strike the right balance with the mesh details and the time for each run. The computational domain created here is 5 car lengths to the front, 15 car lengths to the rear and 4 car lengths to the side. This is the general thumb rule for the size of the computational domain. The mesh size over the surface of the car was made fine (10 mm) in order to capture the surface boundary layer and the volume mesh size i.e., the mesh around the car was made coarse (1000mm). To capture the rear wing upwash and the wake created by the car, a volume refinement in the shape of a cone is done as shown in figure 3. And another volumetric refinement was done around the car in order to capture the flow around the car as shown in figure 4. Figure 5 shows the complete volume mesh of the domain in an isometric view.

An additional local Cartesian coordinate system was created at the center of each wheel in order to set it rotating. The wheels were set rotating at 460 rpm. The floor is set up as moving wall at 30 mph. computational domains define the wall boundary conditions. Figure 6 shows the Computational domain with assigned wall boundary conditions.

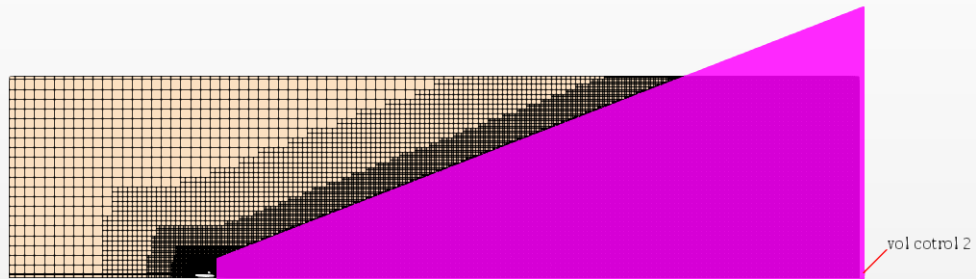


Figure 3: Cone shaped volumetric refinement

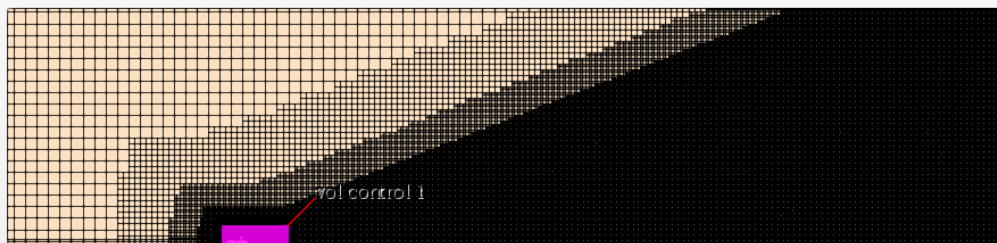


Figure 4: volumetric refinement 1(rectangular box)

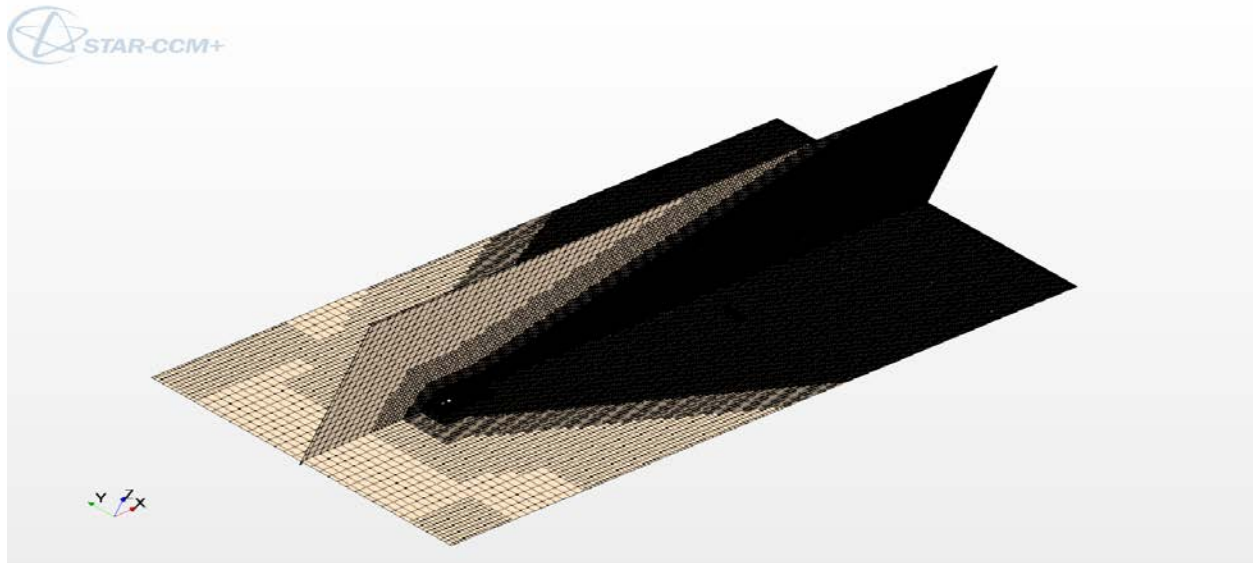


Figure 5: volume mesh in isometric view

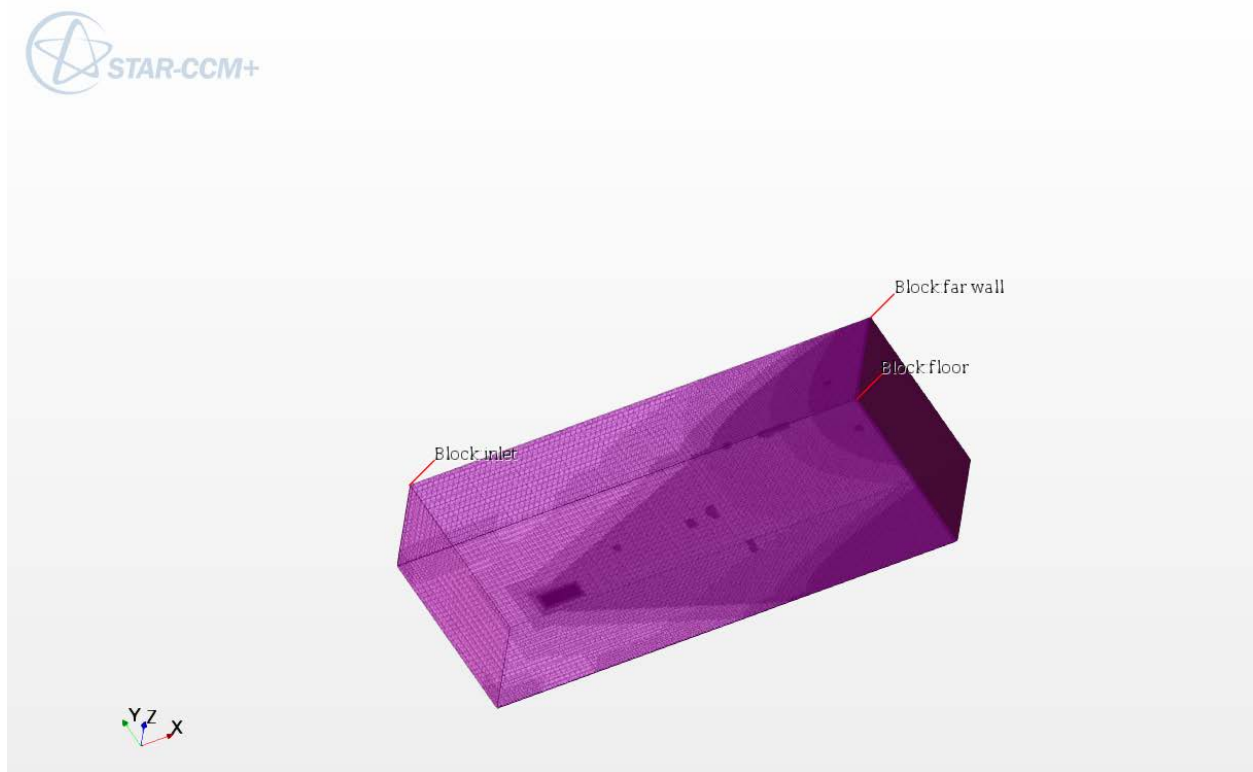


Figure 6: Computational domain showing wall boundary conditions

The solver used here is k-Omega based shear stress transfer (SST) model. The k-Omega 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 (www.arc.vt.edu/ansys_help/cfx_thry).

The model is then allowed to run for several hours until the solution converges. Here the convergence is judged by monitoring the engineering quantity. The convergence is assumed when the engineering quantities has reached unchanged states. If the monitoring quantity is fluctuating greatly, then it suggests that the solution is not converged (steve.cd-adapco.com/articles). The monitoring quantities here are Downforce, Drag, Downforce coefficient and Drag coefficient.

Once the solution is converged, then we move on to post processing section where we generate the engineering values and look at images and plots for further analysis which is discussed in results section.

Half car analysis:

The half car analysis is done on the model if it is symmetrical along the longitudinal central axis. For example if the model does not include the radiator duct and muffler.

This type of analysis is basically done in order to reduce the mesh size and thus we can save computational time. The table below shows the setup of preprocessor, solver and Postprocessor

Preprocessor	Solver	Post processor
<ul style="list-style-type: none"> • Velocity Inlet: 30mph • Pressure outlet • Floor: moving wall condition at 30 mph • Symmetric wall 	<ul style="list-style-type: none"> • 3 dimensional flow 	<ul style="list-style-type: none"> • Downforce • Drag • Drag and downforce coefficient • Front balance
<ul style="list-style-type: none"> • Wheel set rotating at 460 rpm 	<ul style="list-style-type: none"> • K-omega RANS model 	<ul style="list-style-type: none"> • Scalar plots • Vector plots
<ul style="list-style-type: none"> • Mesh type: Hexahedral (trimmer) 	<ul style="list-style-type: none"> • Segregated flow 	<ul style="list-style-type: none"> • Streamlines • Resampled volume

<ul style="list-style-type: none"> • Surface mesh size: 10 mm • Volume mesh size: 1000mm • Volume growth rate: Very slow • Surface growth rate: Very slow • Volumetric control: 50 mm 	<ul style="list-style-type: none"> • Steady state 	<ul style="list-style-type: none"> • Threshold
--	--	---

The figure below shows an example of a symmetrical model along the central axis.

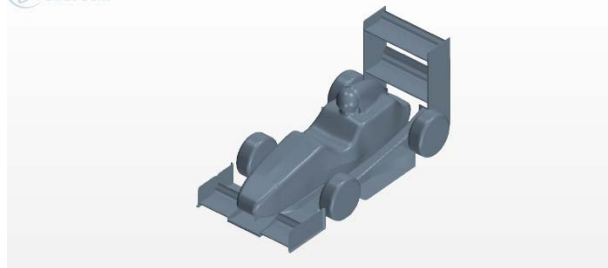


Figure 7: Symmetric car model

Once the model is imported, a computational domain is created which passes along the center of the car as shown in the figure 9. The computational domain used is similar to the full car analysis except that it is exactly half the domain size as compared to computational domain for full car analysis. Figure 9 shows the computational domain for the half car analysis. The grey color is the car and blue color shows the computational domain.



Figure 8: Computational domain

Figure 9 and 10 shows the computational domain with the surface mesh on the half car and the volume mesh. We can observe that it has a volumetric control in the wake region in form of a rectangle. It is again done for the sake of reducing mesh size and saving some computational time. The volumetric control is a fine mesh region.

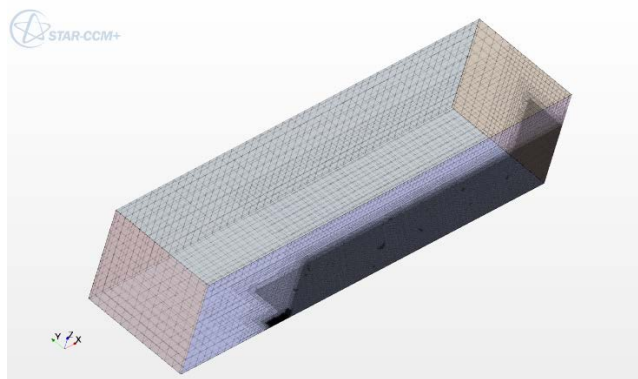


Figure 9: Computational domain with volume mesh

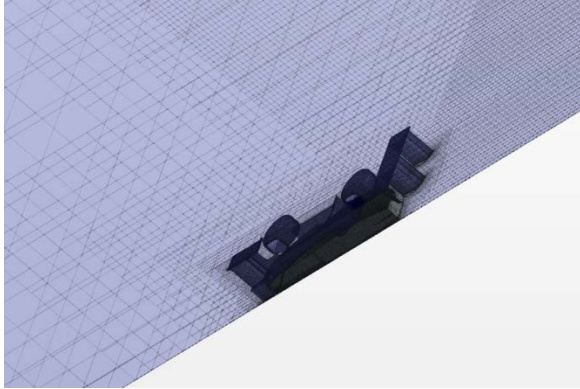


Figure 10: half car surface mesh

Wing analysis:

Apart from half car and full car simulation, there were few CFD runs done on the wings model in order to determine its flow field and to check if there is any flow separation along the wing surface. The table below shows the mesh values and the boundary conditions used. Here instead of k-W turbulence model, we use Spalart-Allmaras model. The Spalart-Allmaras model is a one equation model for turbulence viscosity.

The Spalart-Allmaras model adds a single additional variable for a Spalart-Allmaras viscosity and does not use any wall functions; it solves the entire flow field. The model was originally developed for aerodynamics applications and is advantageous in that it solves for only a single additional variable. This makes it less memory-intensive than the other models that solve the flow field in the buffer layer. Its advantage is that it is quite stable and shows good convergence (www.comsolblogCFD.com). The table below shows the Preprocessor and the solver set up.

Preprocessor	Solver
<p>Hexahedral Mesh</p> <p>Surface Mesh:</p> <ul style="list-style-type: none"> • Maximum: 3mm • Minimum: 3mm 	<p>Spalart-Allmaras Turbulence Model</p>
<p>Volume Mesh:</p> <ul style="list-style-type: none"> • Outside: 500mm • Volume Control: 50mm <p>8 prism layers</p> <p>Boundary conditions:</p> <ul style="list-style-type: none"> • Velocity inlet (30 mph) • Pressure outlet • Far walls 	

Figure 11 shows the one of the wing assembly which was used in this project. Figure 12 shows the Volumetric mesh. The black part indicates a very fine mesh where flow has to be captured (Volume control), and the blue part indicates a coarse mesh (Outside volume mesh). Here the flow capture is of prime importance. Figure 13 shows the prism layers along the surface of airfoil so as to capture the boundary layers and closely monitor the flow separation.

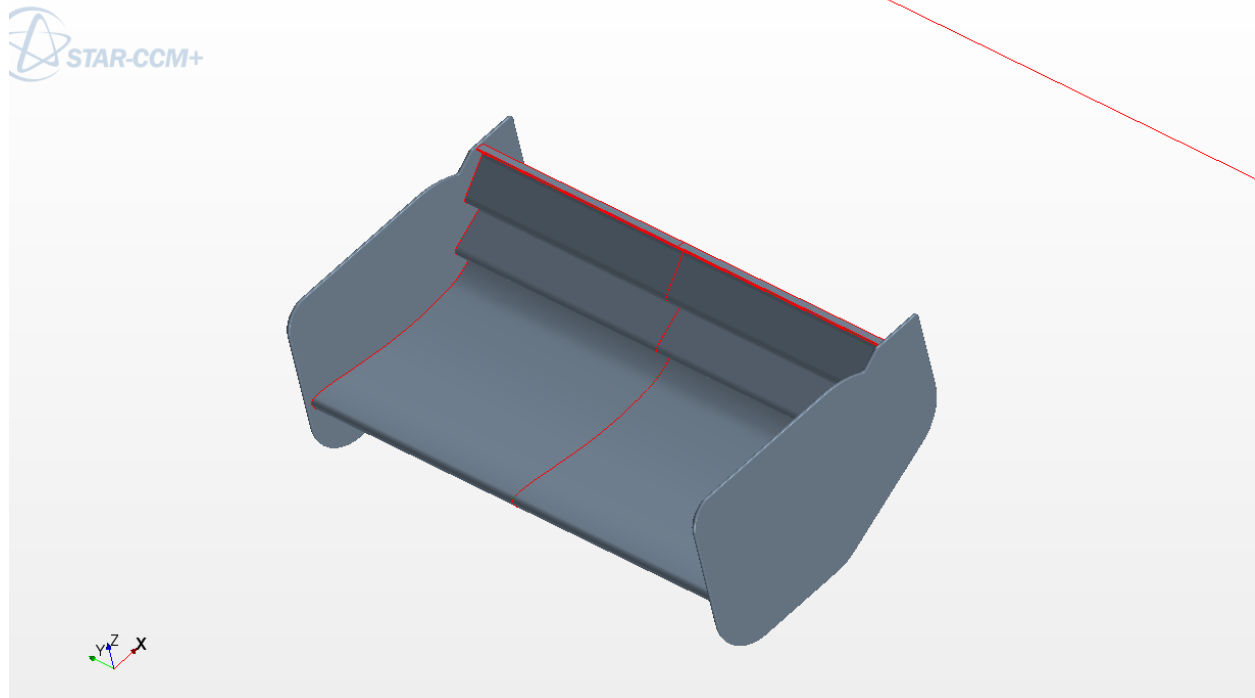


Figure 11: Example of a wing assembly

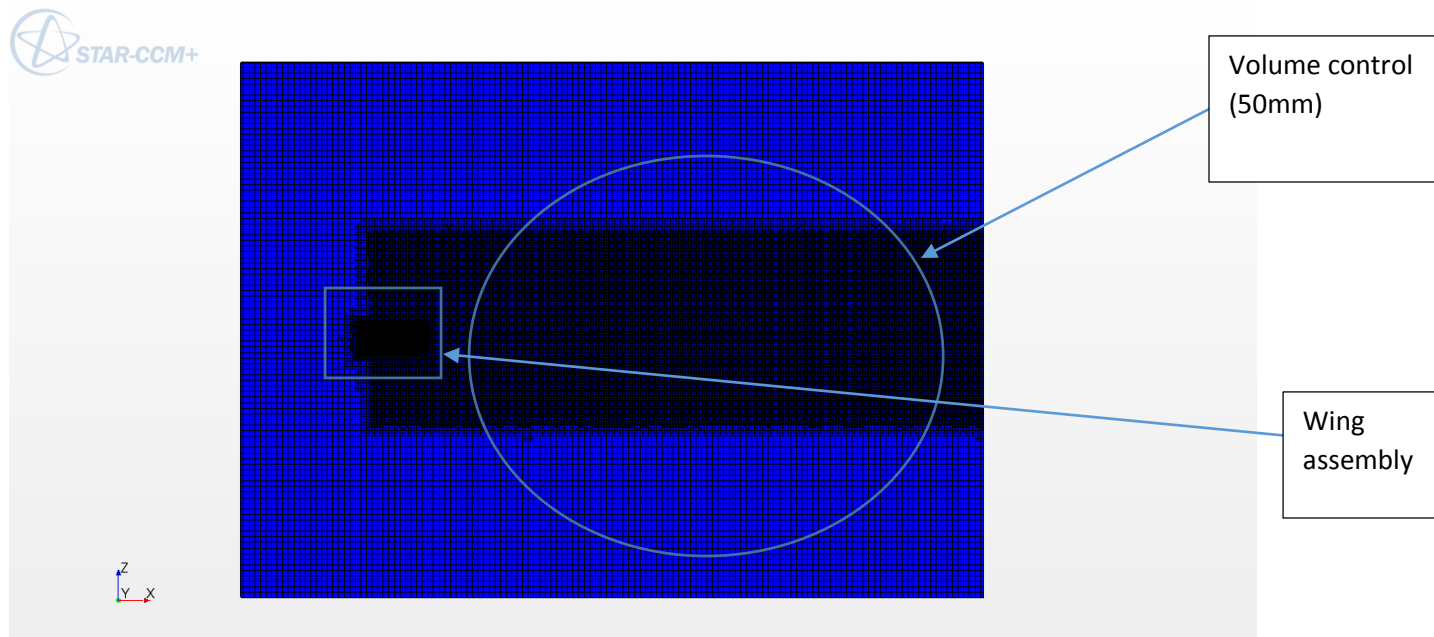


Figure 12: Volume mesh

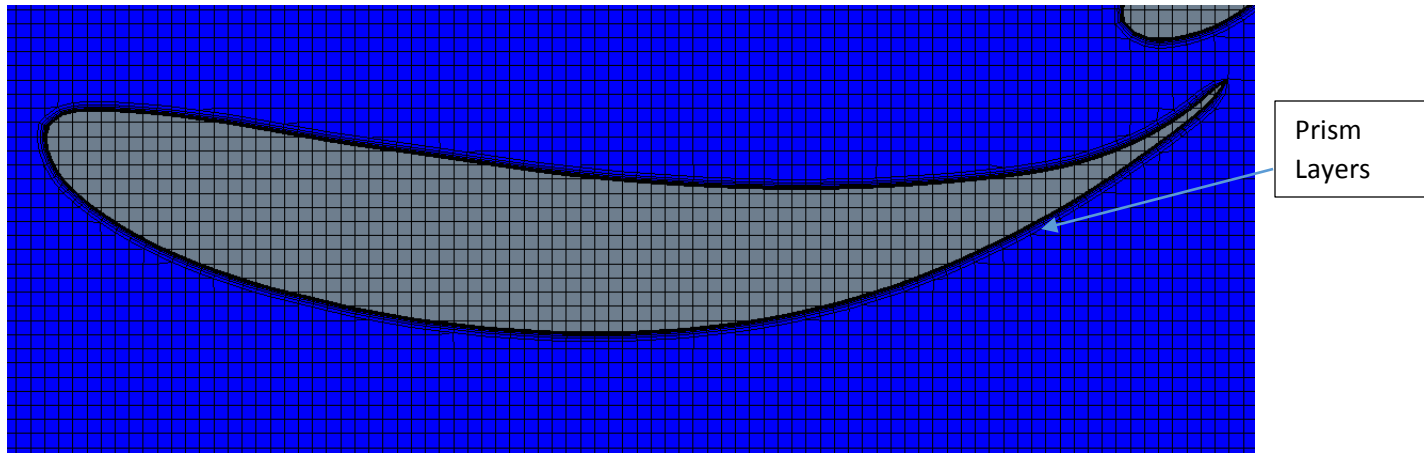


Figure 13: Prism layers to capture the flow around airfoil

Diffuser analysis

In the initial stages of project there were few diffuser analysis done in order to determine the efficient diffuser design for the car. The maximum mesh size for surface mesh was 15mm and minimum was 10mm. Figure 14 shows the surface mesh over one of the diffuser assembly used for our analysis. Coarser mesh on the image refers to maximum mesh size whereas finer mesh which is generally on the edges of diffuser and tires refers to minimum mesh size.

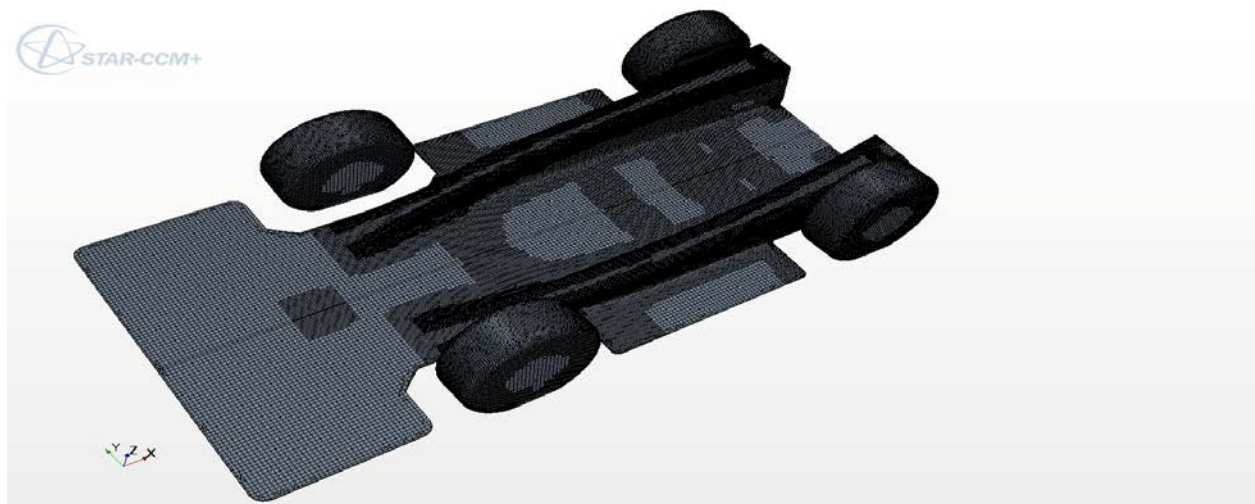


Figure 14: Surface mesh

Figure 15 shows the volumetric mesh. The black part indicates a very fine mesh which refers to minimum mesh and the blue part which is coarser mesh refers to maximum mesh size. The transition from minimum mesh size to the maximum mesh size is smooth and uniform. The minimum mesh is incorporated in order to capture the airflow along the diffuser surface and regions of adverse pressure gradient around the diffuser and also in the wake region. The coarse meshing has been set up where the significance of flow characteristics is of less importance. Once the meshing is done, we input the boundary conditions along the walls of computational domain.

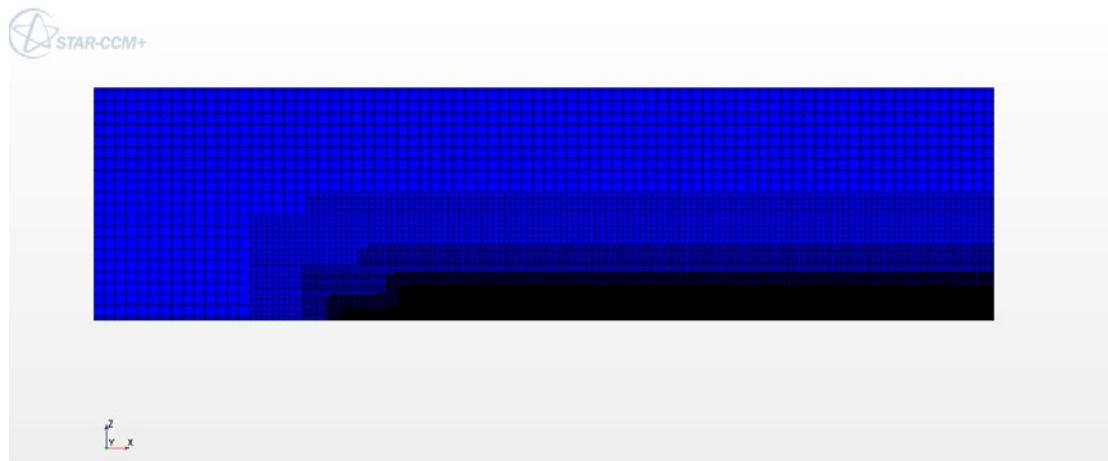


Figure 15: Volumetric mesh

The table below shows the set up for preprocessor and the solver. Here the computational domain chosen was diffuser length 3 diffuser length to the front, 8 to the rear and 3 to the sides. The volumetric refinement was fine as compare to other simulation.

Preprocessor	Solver
Trimmer Mesh Surface Mesh: <ul style="list-style-type: none"> Maximum: 15mm Minimum: 10mm 	K ω Turbulence Model
Volume Mesh: <ul style="list-style-type: none"> Outside: 200mm Volume Control: 50mm 	

Radiator set up

To model the radiator, we consider it as a block of honeycomb material, much like a bunch of drinking straws that are glued together. This porous medium is situated in a section of the duct that is oriented at 45 degrees to the x axis of the local coordinate system as shown in the in figure

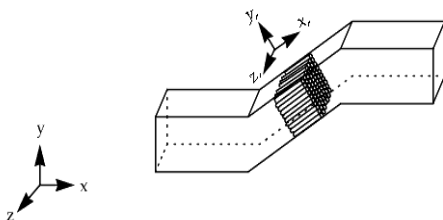


Figure 16: Radiator modelling

- Measurement of the honeycomb material in a wind tunnel (air as a working fluid) has shown that it results in a pressure drop of 70 pa per meter of the material when the velocity is 1 m/s. In addition, for the range of velocity of interest, the pressure drop varies linearly with velocity.(*User Guide STAR-CCM+ Version 8.06*. 2013)

Since the porous resistance is a linear function of flow velocity, only the viscous resistance tensor is specified. In principle coordinate, the primary coefficient is 70. To restrict the flow in the direction perpendicular to the primary direction, we choose a resistance coefficient that is in 2-3 orders of magnitude larger than primary coefficient, for example 10000. Choosing a larger resistance does not affect the flow but can adversely affect the convergence characteristic. Therefore the viscous resistance tensor in principal coordinate is chosen as shown below.

$$\begin{bmatrix} 70 & 0 & 0 \\ 0 & 10000 & 0 \\ 0 & 0 & 10000 \end{bmatrix}$$

The figure below shows how the radiator has been set up inside the radiator duct. So the radiator block is modelled as a porous medium. The flow is expected to experience a pressure drop as it flows through the radiator block.

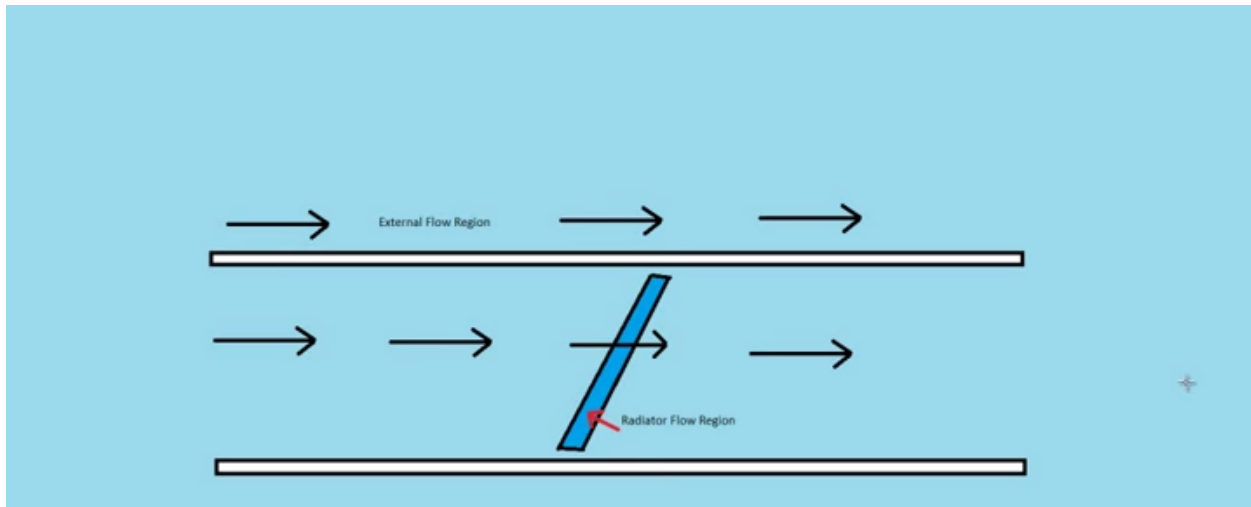


Figure 17: Radiator set up

Aero balance calculation

The aerodynamic balance or the center of pressure is calculated as shown below. Figure 18 shows an example of a race car and how we can calculate COP for that.

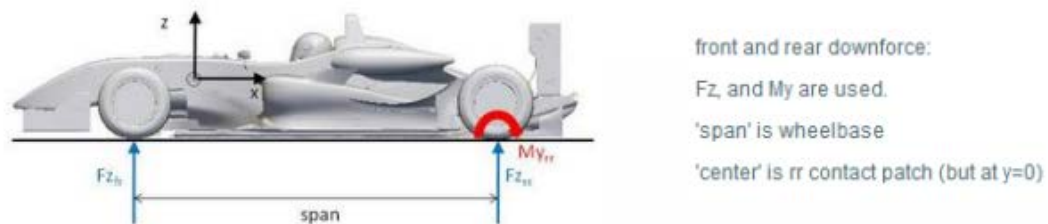


Figure 18: Example to depict the COP calculation

Step 1: We begin by obtaining F_x , F_z and My_0

Where F_x is the force along X axis,

F_z is the force along Z direction

My_0 is the moment about the origin

For all the COP calculation in our project, the origin was set at the center of the front axle of the car.

Step 2: We then calculate the moment (M_y) about the center of the rear tire contact patch by using the equation shown below for moment translation:

$$M_y = M_{y0} + dxF_z - dzF_x$$

Where dx and dz are the distances from the origin to the contact patch parallel and normal to the ground plane.

Step 3: Front downforce is then calculated by dividing the contact patch moment by the wheel base of the car, L

$$F_{\text{front}} = M_y/L$$

Step 4: Finally balance is the ratio of the front downforce to the total downforce:

$$\text{Balance} = F_{\text{front}}/F_z$$

And rear downforce is:

$$F_{\text{rear}} = F_z - F_{\text{front}}$$

Results and Findings

2015 FSAE design evolution

The figure below shows the first basic concept the team came up with the advice from faculty advisor. The idea was to have a diffuser tunnel starting right from the forward of the front tires and have a multi elements rear wings. A lot of testing had to be done on this design concept so as to implement it on the car as will be explained in the following design iterations. This a basic starting point for the design process. Figure 20 shows the basic idea for our design. Though it was not yet designed accurately to fit around the chassis, but a basic idea could be established nevertheless.

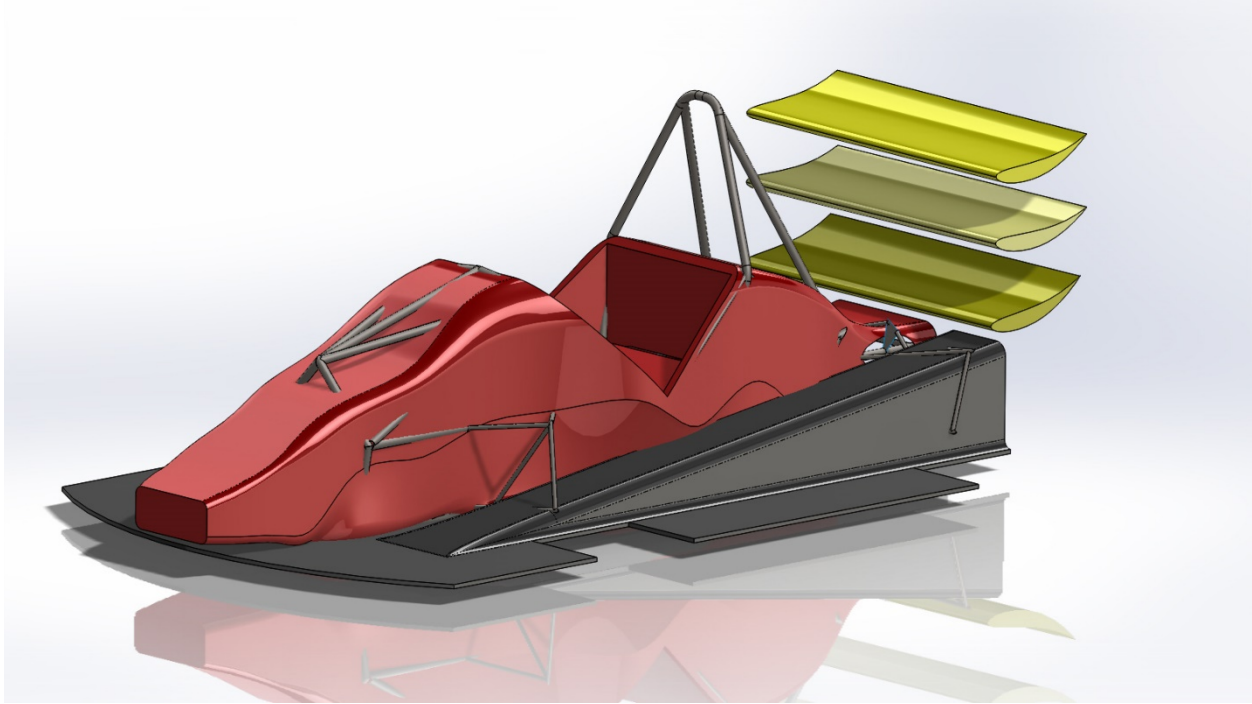


Figure 19: Basic idea for the 2015 aero package

Summary Table

The following is a summary table for the CFD simulation done. There were 7 design iterations done. Design changes were made after each iteration and the simulation were run. Some of the simulations converged at 500 iteration and 700 iteration and others at 1000 iterations. All the simulation was run at 30 mph which is average velocity of the FSAE car at the competition. Drag and downforce are summarized. C_z refers to the downforce coefficient, C_x refers to drag Coefficient. The ratio (C_z/C_x) gives the efficiency of the aero package. The COP is measured in terms of front balance i.e. the percentage of downforce acting on the front axle.

Iteration	Configuration	Design change	Number of cells	Convergence	speed (mph)	Frontal area (m ²)	Downforce (lbs.)	Drag (lbs.)	Cz	Cx	Effeciency (Cz/Cx)	Front Balance (%)
1	Half car analysis	1. Nosecone 2. Diffuser 3. 2 element rear wing (NACA 9514)	11 million	Fully Converged at 500 iteration	30	—	—	—	—	—	—	—
2	Half car analysis	1. Nosecone 2. Diffuser 3. 3 element rear wing(NACA 9514) 4. Engine cover	13 million	Fully converged at 700 iteration	30	0.945	34	18.66	1.43	0.79	1.81	—
3	Half car analysis	1. Nosecone 2. Diffuser 3. Rear wing assembly(Indycar): 3 elements on bottom 2 elements on the top 4. Addition of Gurney Flaps	13.5 million	Fully Converged at 1000 iteration	30	0.967	41	21	1.74	0.84	2.07	17.5
4	Half car analysis	1. 2 elements Front wings(Indycar) With AoA 5 deg 2. Rear wing assembly: 3 element on the top and 3 element on the bottom 3. Gap between helmet and engine cover closed	15 million	Fully converged at 1000 iteration	30	0.997	63	23	2.09	0.93	2.25	30
5	Half car analysis	1. 2 element front wing with AoA 7 deg 2. Rear wing assembly: (2 element on the top and 3 on the bottom) 3. Airfoil shaped diffuser leading edge 4. center part of the wing assembly flattened	14 million	Fully converged at 1000 iteration	30	0.988	58	21	2.37	0.85	2.79	47.2
6	Full car analysis	1. 2 element front wing(AoA 7 deg) 2. Radiator duct 3. Muffler 4. Rear wing assembly: 3 element on the top and 2 on the bottom	25 million	fully converged at 1000 iteration	30	1.13	76.4	28.7	2.73	1.02	2.68	50.7
7	Full car analysis	1. 2 element front wing(AoA 7 deg) 2. Radiator duct 3. Muffler 4. Rear wing assembly: 3 element on the top and 3 on the bottom	25.5 million	Fully converged at 1000 iteration	30	1.23	84.3	36	2.74	1.17	2.34	48.8

Figure 20: CFD Summary table

Design Iteration 1:

The team came up with a general design just to establish a real estate. This design was done in order to fit the parts around the chassis which was already built by team. The design comprised of a nosecone, diffuser and a 2 element rear wings (NACA 9514). The details of NACA profile is mentioned in literature review section. Figure 21 and 22 shows the model for the design 1 iteration.

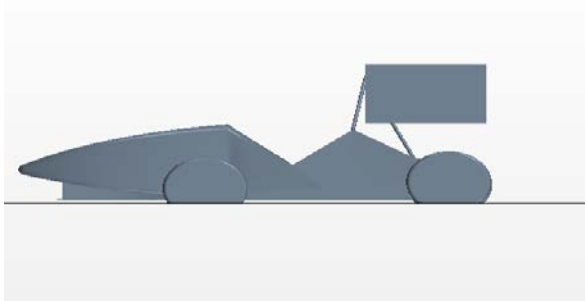


Figure 22: side view



Figure 21: Front view

A CFD simulation was done on this model to see the airflow around the car. Here the main focus was just on the airflow characteristic and not on the aerodynamic numbers.

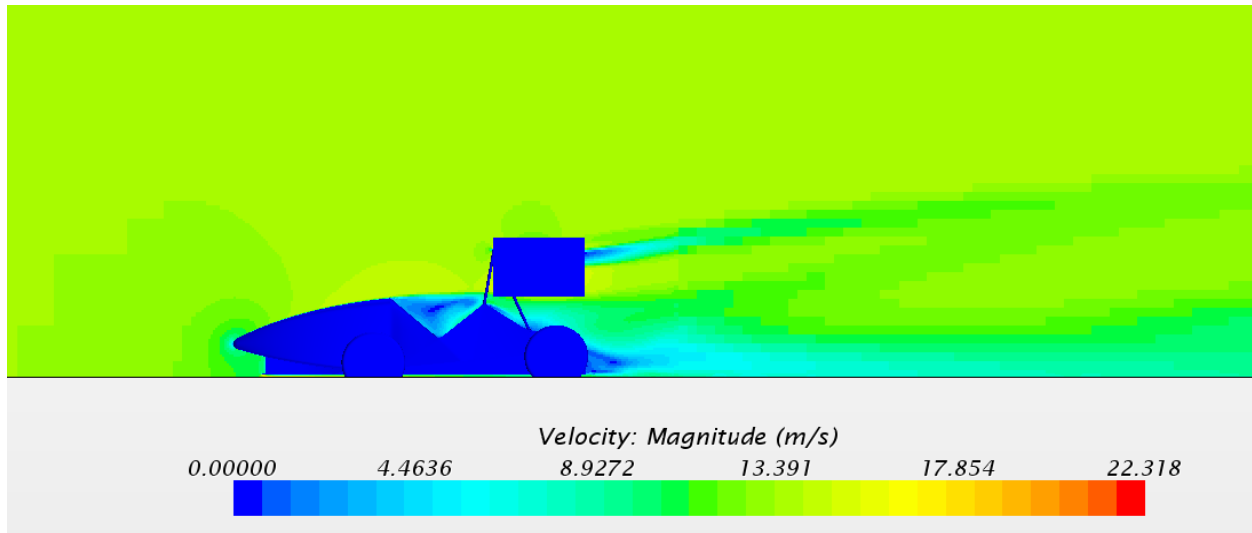


Figure 23: Scalar plot: Velocity magnitude

Figure 23 shows the scalar field (Velocity magnitude) along the center section of the car. A closer look at the CFD plot shows that there is a flow separation on the rear wings (figure 24). The next step was to check what would be the best angle of attack (AoA) at which the rear wing can be placed so that there is no flow separation.

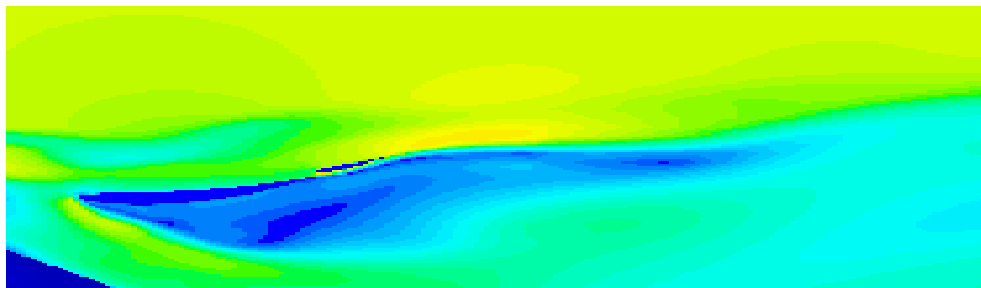


Figure 24: rear wing flow separation

Few CFD runs were done on NACA 9514 airfoil at 30 Mph to determine the optimal angle of attack (figure 25). Here the blue color region indicates turbulence/ flow separation. From the following figure we can see there is a large flow separation at 14 and 10 deg. Flow separation reduces at 10 deg. And the flow is completely attached at 8 and 6 deg. Thus, 8 degrees was the optimal angle for a single element airfoil. We could then go on to increase the AoA by adding more elements.

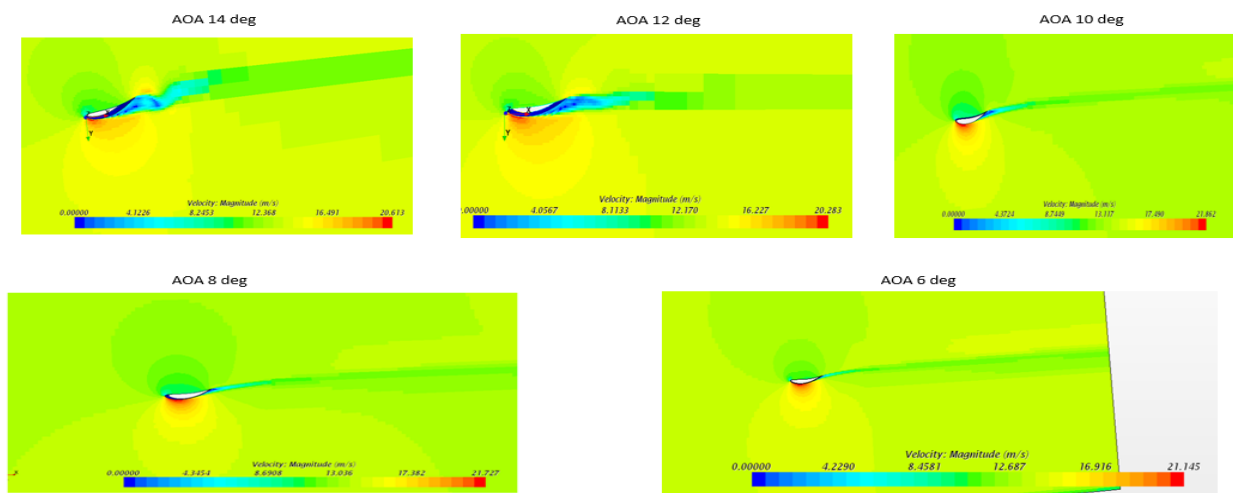


Figure 25: CFD runs on NACA 9514 at different AOA

Diffuser Sweep

A CFD study was done in order to find the optimum diffuser angle for our car. A diffuser set up done on the CFD model as described in the methodology section. Simulation was run at 30 mph and 1" ride height.

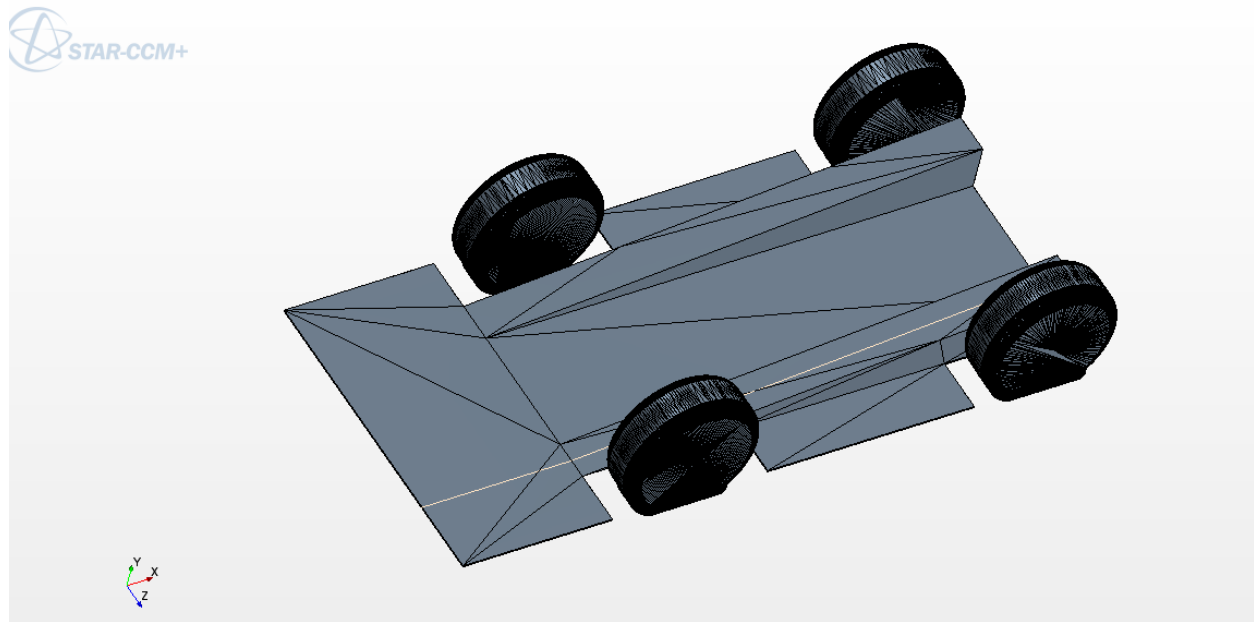


Figure 26: Basic diffuser shape design

The CFD simulation was done at four different diffuser/ramp angles i.e. at 7, 11, 15 and 17 degrees. After the simulation was run, a graph was plotted for downforce, drag and efficiency (L/D) against different ramp angles as shown below. Here the efficiency is referenced to the ratio of downforce to that of drag. The graph suggested that there was decrease in downforce number at 15 deg., which suggests that the diffuser stalled close to 15 degrees. According to the theory, the diffuser is expected to stall at 10 deg at 1" ride height (www.formula1dictionary.net). This CFD study was considered as very optimistic Therefore, an intermediate value of 12 deg. was chosen as the ramp angles.

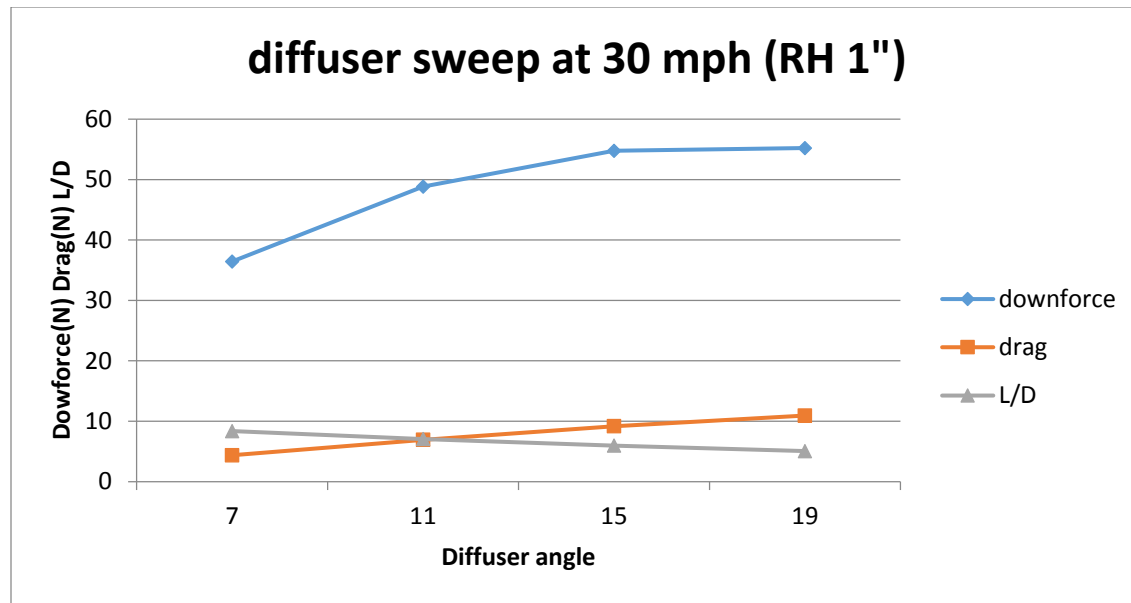


Figure 27: Diffuser sweep

Design iteration 2:

The next iteration comprised of a new diffuser which was shaped around to fit the chassis and at 12 degrees ramp angle which was found from the previous study. The diffuser design which fits around the chassis is shown in the figure 28.

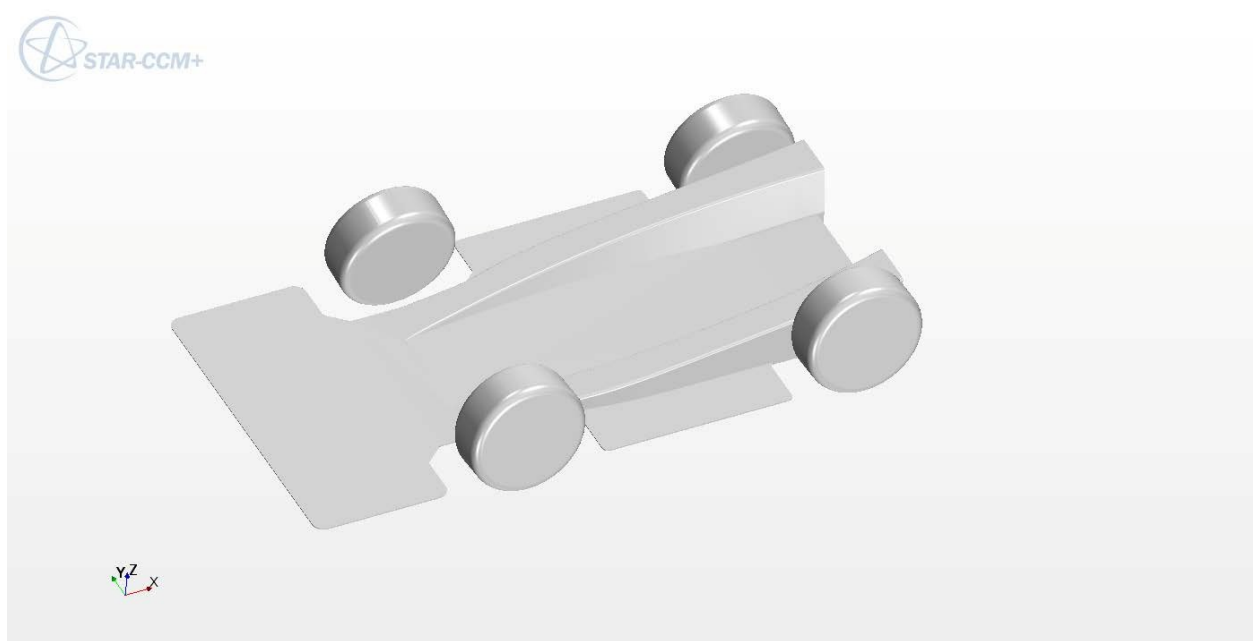


Figure 28: Diffuser design to hug around the chassis

The two element wings were then changed to 3 elements NACA 9514 to increase the downforce numbers. And also an attempt was made to improve the diffuser efficiency by adding a wing just above the diffuser exit to improve diffuser pumping. Diffuser Pumping refers to the increased cross-section area over the diffuser length, which can be used to increase the flow rate through diffuser because of pressure potential. As the ratio of the inlet to outlet area becomes increasingly greater, this generates greater pressure recovery that, due to the base pressure remaining constant will increasingly depress the base pressure at the inlet. The diffuser acts to reduce the underbody pressure due to the expansion resulting in increased flow rate under the body. This scavenging both produces a lower pressure area under the car and also acts to reduce the boundary layer. (<http://formula1-dictionary.net/diffuser.html>)

Thus the new design comprised of a 3 element rear wing and a single element over the diffuser exit as shown in figure 29.

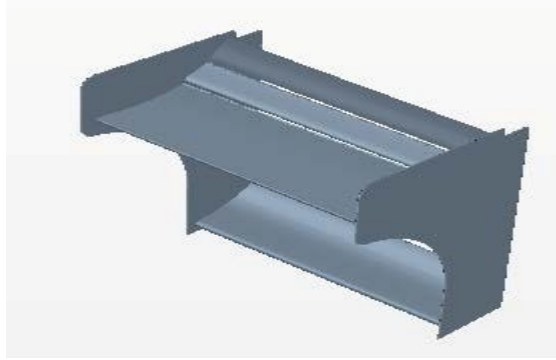


Figure 29: Design 2: Rear wing package

The complete package for design iteration 2 comprised of a modified nose cone, new diffuser, an engine cover and the rear wing package as shown in the figure below

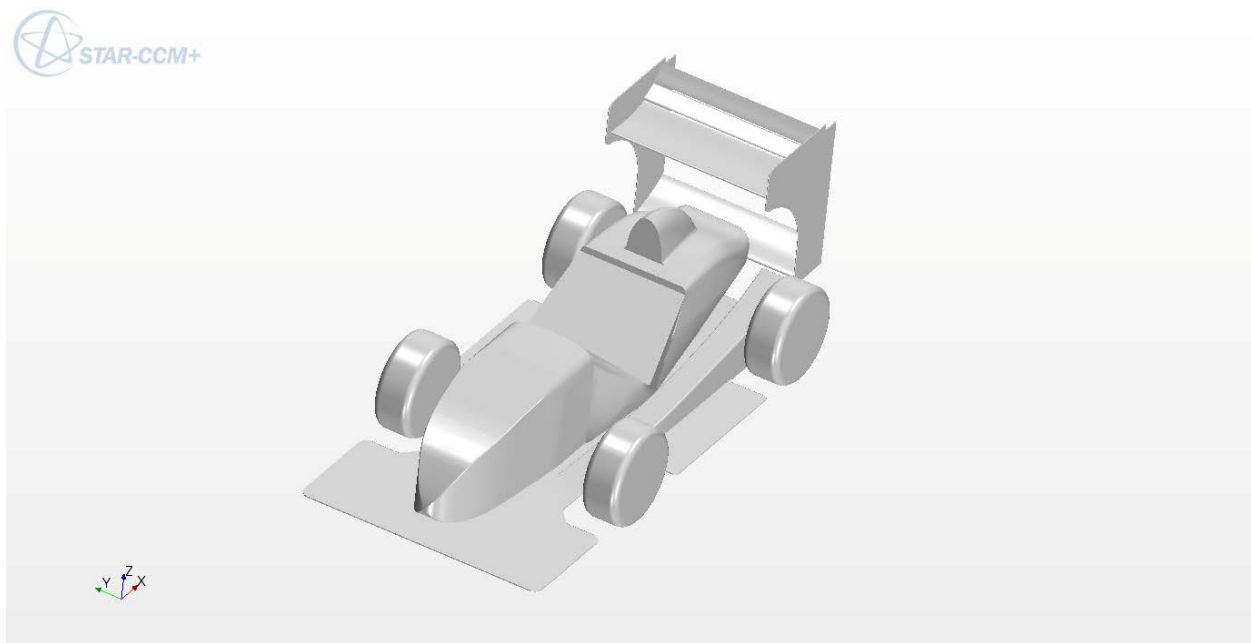


Figure 30: Design iteration 2

CFD results

A half car simulation was done on this model and CFD results are shown in the table below.

<u>Parameters</u>	<u>Values</u>
Downforce	34 lbs.
Drag	18.66 lbs.
Frontal area	0.945 m ²
Cd	0.79
CL	1.43
Efficiency	1.81

Now we have established a baseline from which we can to improve our aerodynamic numbers.

Figure 31 shows the scalar plot for the static pressure on the surface of the car. A few streamline were added in order to visualize the flow characteristic around the car.

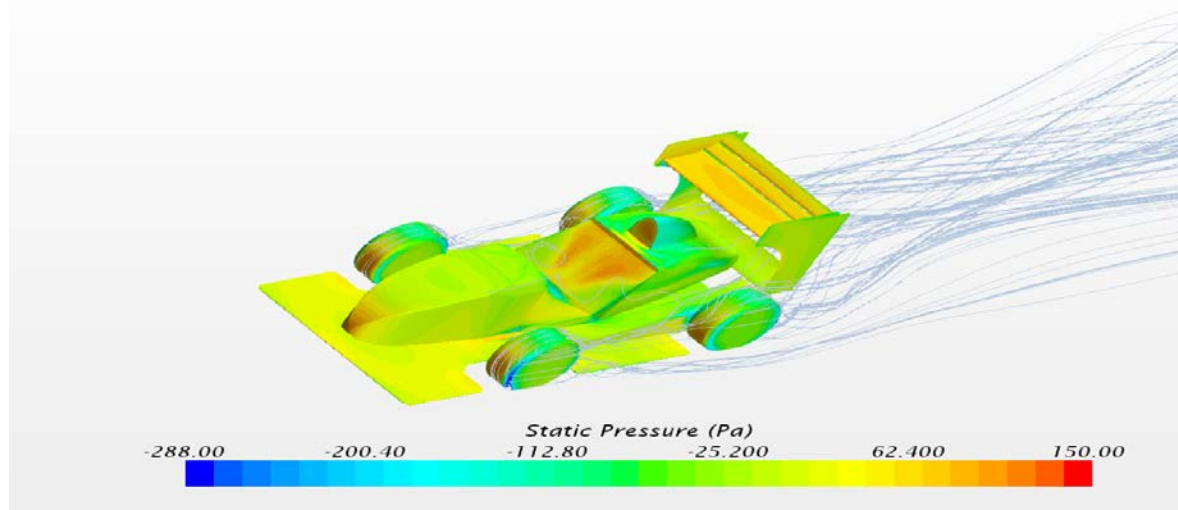


Figure 31: Scalar plot: Static pressure with streamlines

Findings:

1. The wings are not working as efficiently as it should. We found the rear wings stall. The bottom element stalls a lot and does not produce diffuser pumping. We can observe from the figure below that there is small flow detachment on the third element of upper rear wing and single element bottom wing stalls significantly.

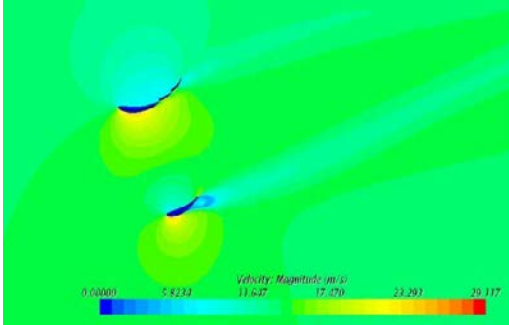


Figure 32: Velocity plot showing rear wing stall

2. The CFD numbers are good compared to other FSAE car with good aerodynamic package. There is room for further improvement. We have established a starting point for the development process.

Design iteration 3

From the findings of design iteration 2, the team with consultation of our faculty advisor decided to go against designing our own wing element. Instead we decided to get the multi element wing profiles from one of the Indy cars team. The wing profiles were built for high velocity application. But the FSAE car is a low velocity car. The average velocity of FSAE car is 30 mph. Hence, we had to do few CFD testing in order to implement Indy car wing profiles on our FSAE car.

Indy car airfoil Study

Figure 33 shows the airfoil shape of the indy car. It consists of 3 elements. It is designed to produce high downforce numbers.

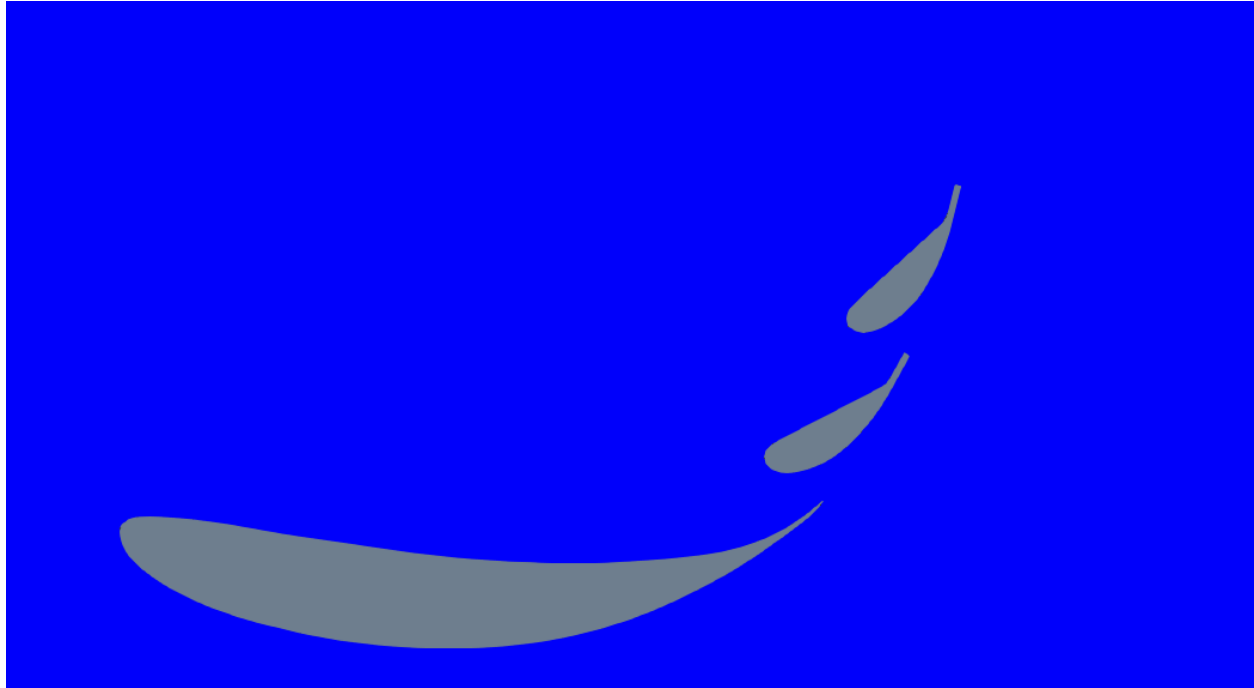


Figure 33: Indycar wing profile

A CFD run was done on the above wing profile in order to check for flow separation along the surface. The simulation was run at 30 mph.

Once the CFD simulation was complete, scalar and vector fields were plotted in order to visualize the flow around the airfoils. Figure 34 and 35 shows scalar and vector field of the wings elements.

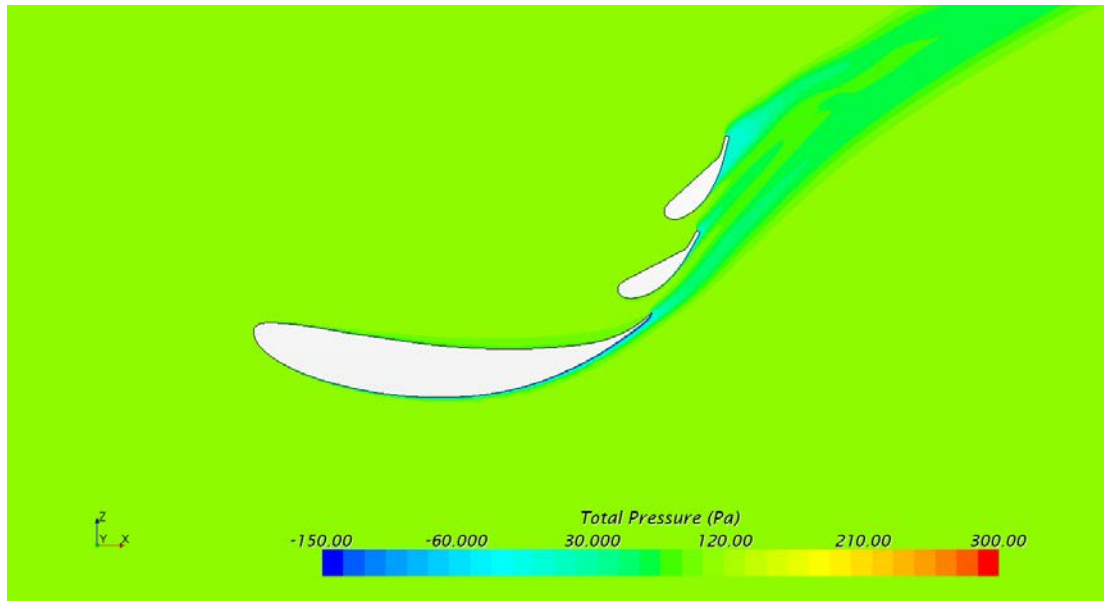


Figure 34: Scalar pressure plot

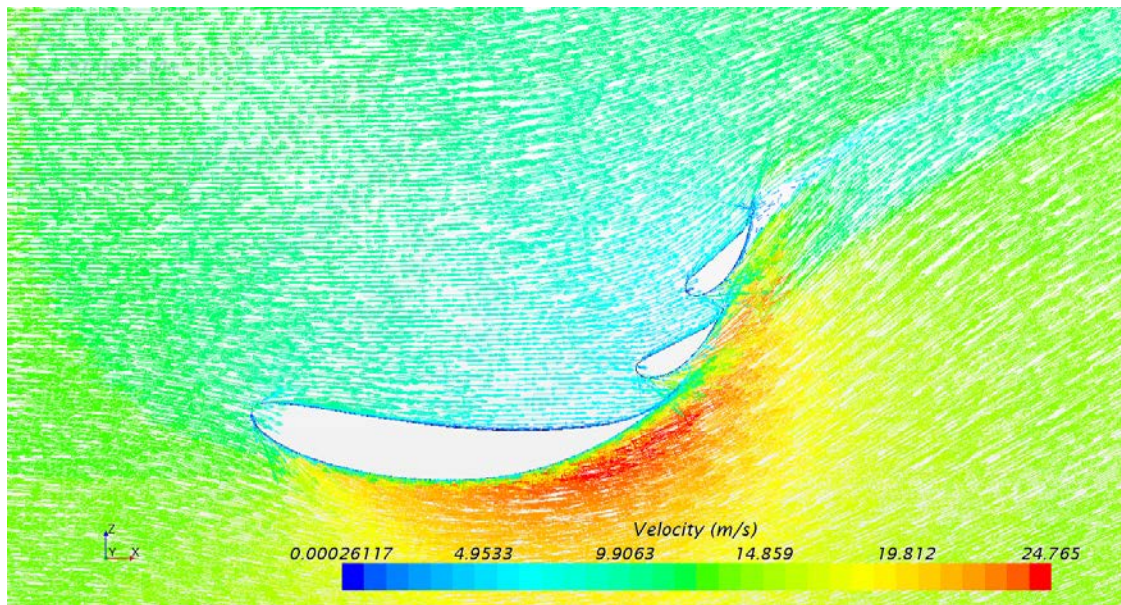


Figure 35: Velocity vector

From the above figure we can see that there is a small amount of flow separation on the top element (third element) of the wing assembly. Figure 36 shows a close up view of the velocity vector on the top element. It clearly shows the flow separation.

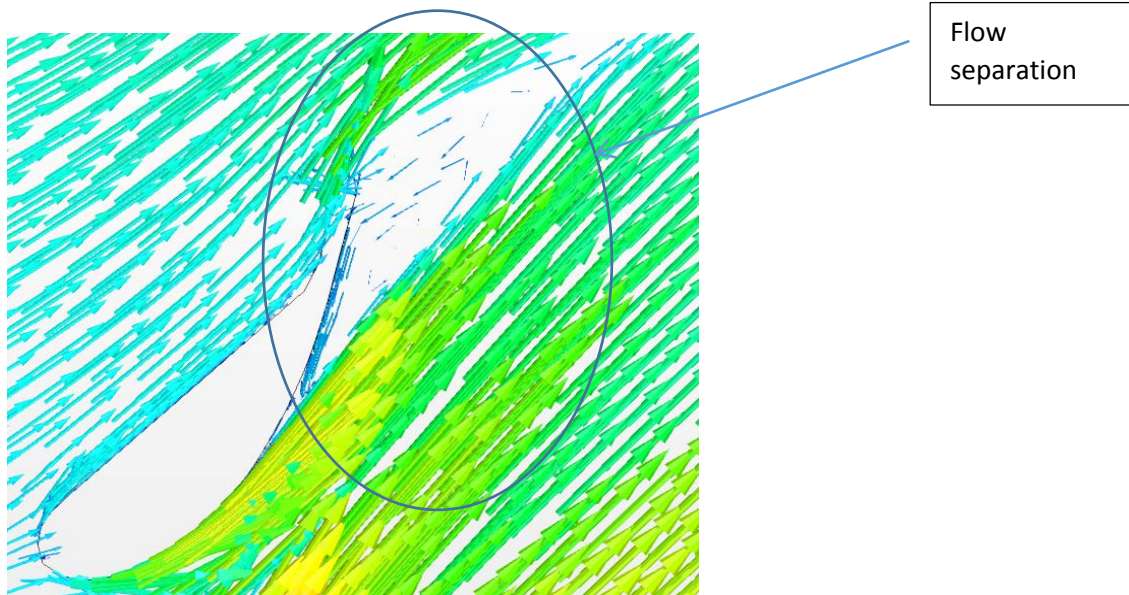


Figure 36: Velocity vectors showing flow separation

To fix this issue, we incorporated a Gurney Flap on this element. The Gurney Flap (or wicker bill) is a small tab projecting from the trailing edge of a wing. Typically it is set at a right angle to the pressure side surface of the airfoil, and projects 1% to 2% of the wing chord. This trailing edge device can improve the performance of a simple airfoil to nearly the same level as a complex high-performance (Giguere, Lemay, Dumas (1995)). Through the proper use of Gurney Flaps, the aerodynamic performance of a simple design, easy-to-build airfoil can be made practically as well as those of a modern, high performance, complex design

We incorporated a 1/2" Gurney Flap on the top of second element in order to improve the performance of the airfoil. We then run a CFD with Gurney and plotted the scalar and vector as shown in the figure and there was an improvement in the flow field. The flow detachment has improved as we can see in the figure 37, 38 and 39.

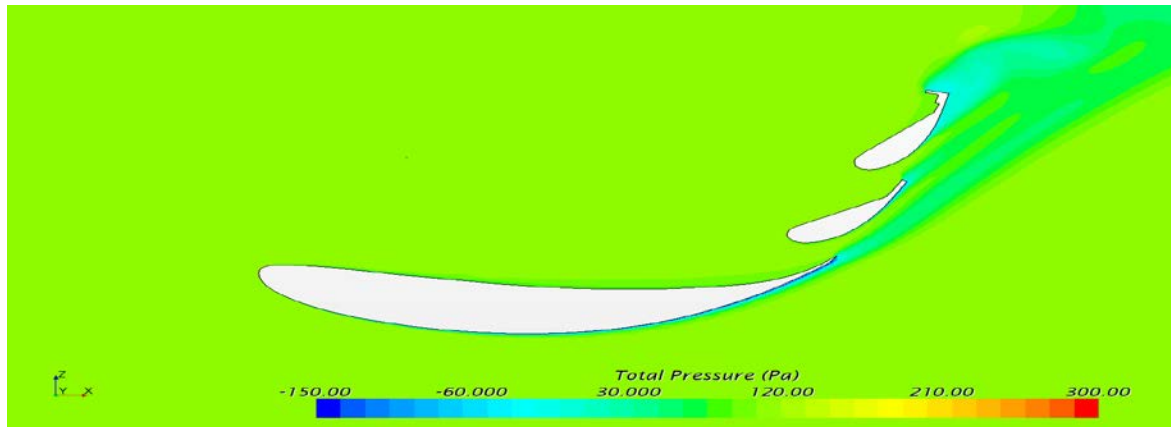


Figure 37: Scalar pressure plot (With gurney)

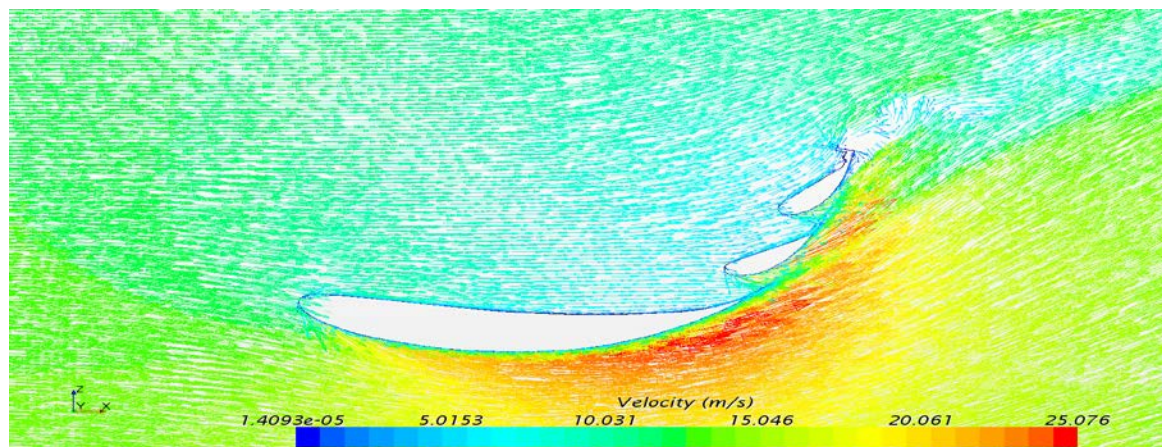


Figure 38: Velocity vector plot (With Gurney)

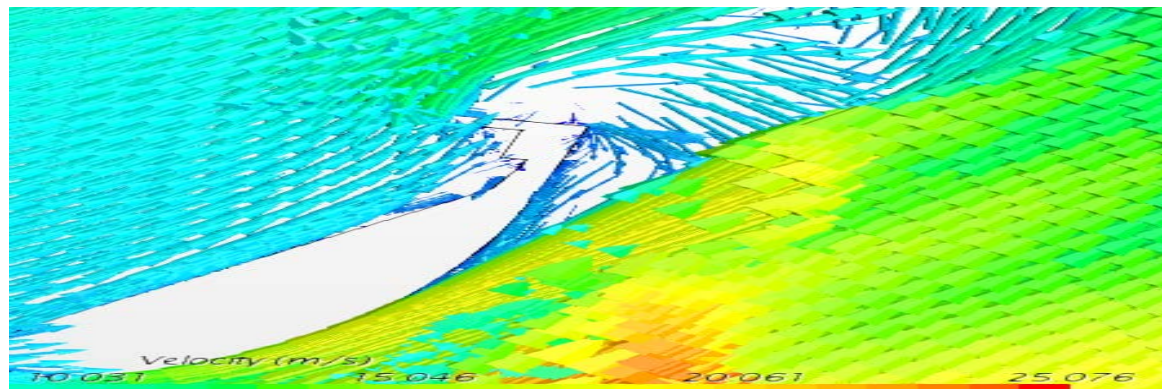


Figure 39: velocity vector depicting the reduced flow separation due to addition of gurney flap

Thus it was evident from the above CFD images that adding a gurney helped in reducing the flow separation on the third element. As a result, the downforce number and the downforce coefficient number also improved.

Car model 3

Based on the above study done on the wings, a new car was designed with these wings (Indycar profiled) on it. To get more realistic flow around the car, a driver model was added. This car model consisted of a diffuser, new nose cone, engine cover and rear wings as aero components.

The rear wings consisted of 3 elements on the bottom positioned at just above diffuser exit and 2 elements on the top.

The CAD model is shown in the figure 40

CCM+

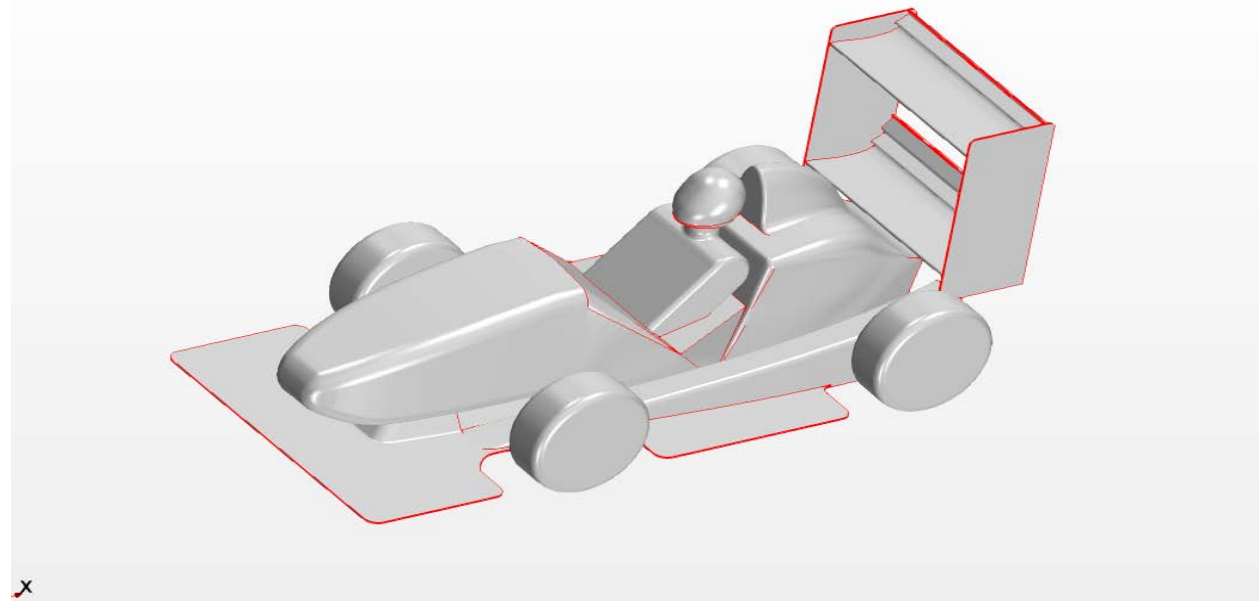


Figure 39: Car model 3

A CFD analysis was done on this model to determine the Drag coefficient (C_x), Downforce coefficient (C_z), downforce and drag. From here on we were also interested in determining the balance of the car or in technical terms center of pressure (COP).

The aerodynamic numbers are as follows:

<u>Parameters</u>	<u>Values</u>
Downforce	41 lbs.
Drag	21 lbs.
Frontal area	0.967 m ²
Cd	0.84
CL	1.74
Efficiency	2.07
Front Axle balance	17.5 %

Findings:

1. Improvement in downforce number from previous iteration.
2. From the above table we can observe that the balance is mostly rearwards.

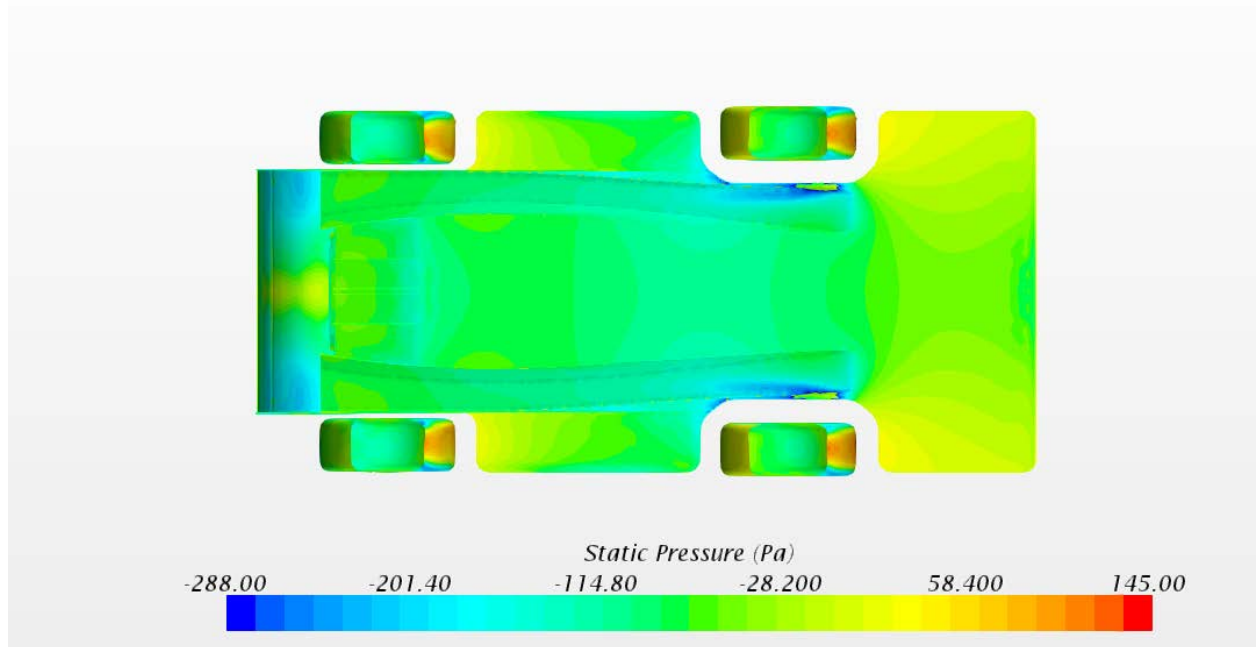


Figure 40: Static pressure Scalar field of underbody

3. We can see more blue color spots on the bottom of rear wings (Except the center part) which indicates low pressure, thus there is large downforce produced on the rear part of the car.
4. **Diffuser pumping:** we can observe more blue color spots on the rear part of the diffuser. This is due to a phenomenon called diffuser pumping. The rear wings have a huge impact on the underbody region. There is an interaction between the low pressure created under the suction side of the rear wing, and the airflow in the diffuser at the back of the underbody (Mcbeath, 1998). The rear wing has a profound beneficial effect on the under car pressure. The rear wing's suction effect amplifies the underbody's pressure reduction (Mcbeath, 1998).

5. **Need for an aerodynamic Headrest:** After a thorough analysis of the airflow around the car, we found that the center part of the rear wing was not working effectively. This was due to a large wake or turbulence created due to the gap between engine cover and helmet as shown in figure 42.

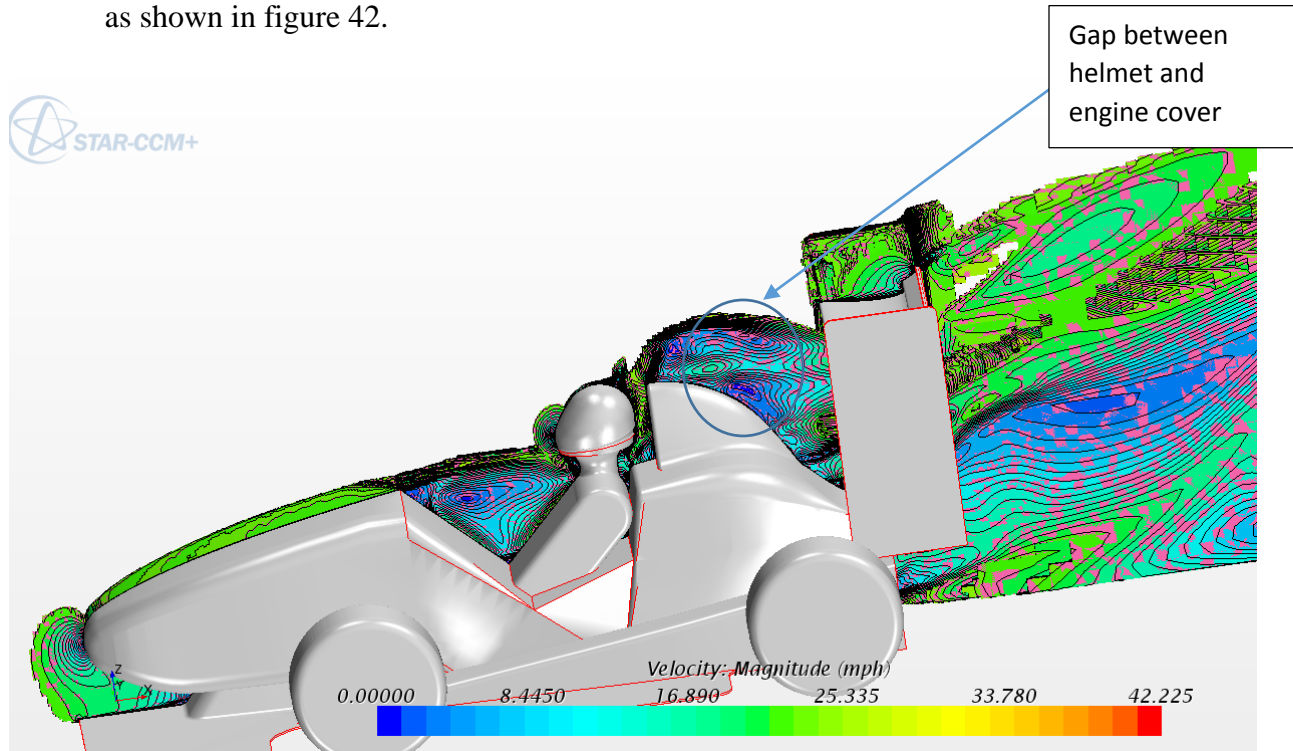


Figure 41: Velocity threshold showing large wake on back of driver's helmet

A closer look at the wake is shown in figure 43. We can observe from this figure that the large low pressure air is striking the top face of the bottom wing assembly; therefore, reducing the performance of the wing.

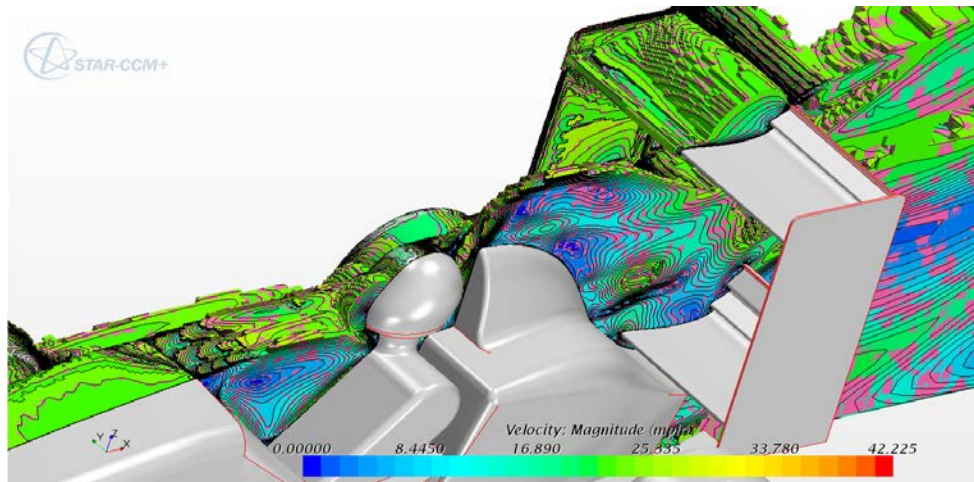


Figure 42: Velocity threshold showing large low velocity air striking the rear wings

Therefore, from this design iteration we found that the downforce number has certainly increased due to implementation of new set of wings. Form the CFD analysis it is found that the balance is mostly rearwards, hence there is a need to add front downforce for good aerodynamic balance.

Also there is a need for an aerodynamic headrest in order to improve the rear wing performance.

And also for the simplification in CFD analysis, the gap between the helmet and engine cover should be closed.

Design iteration 4:

To improve the front downforce, we incorporated a 2 element front wing as shown in figure 44.

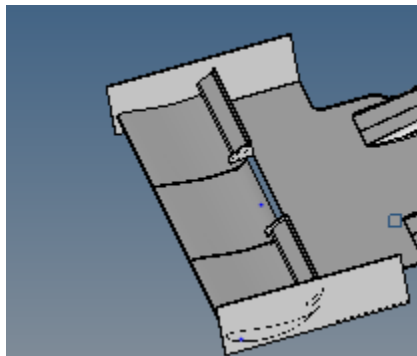


Figure 43: Front wing assembly

Initially the front wing assembly was placed at 5 degrees AoA.

And to deal with the headrest issue, we decided to close the gap between helmet and engine cover as shown in the figure below in figure 45.

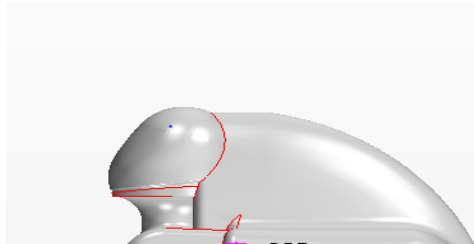


Figure 44: Gap between helmet and engine cover closed

Half car CFD simulation was done and results are as follows:

<u>Parameters</u>	<u>Values</u>
Downforce	63 lbs.
Drag	23 lbs.
Frontal area	0.997 m ²
Front Axle balance	30 %

Findings:

1. Front wing is working well so far. It has added some front downforce. We get good low pressure spots (Blue color) under the front wing(Figure 46).

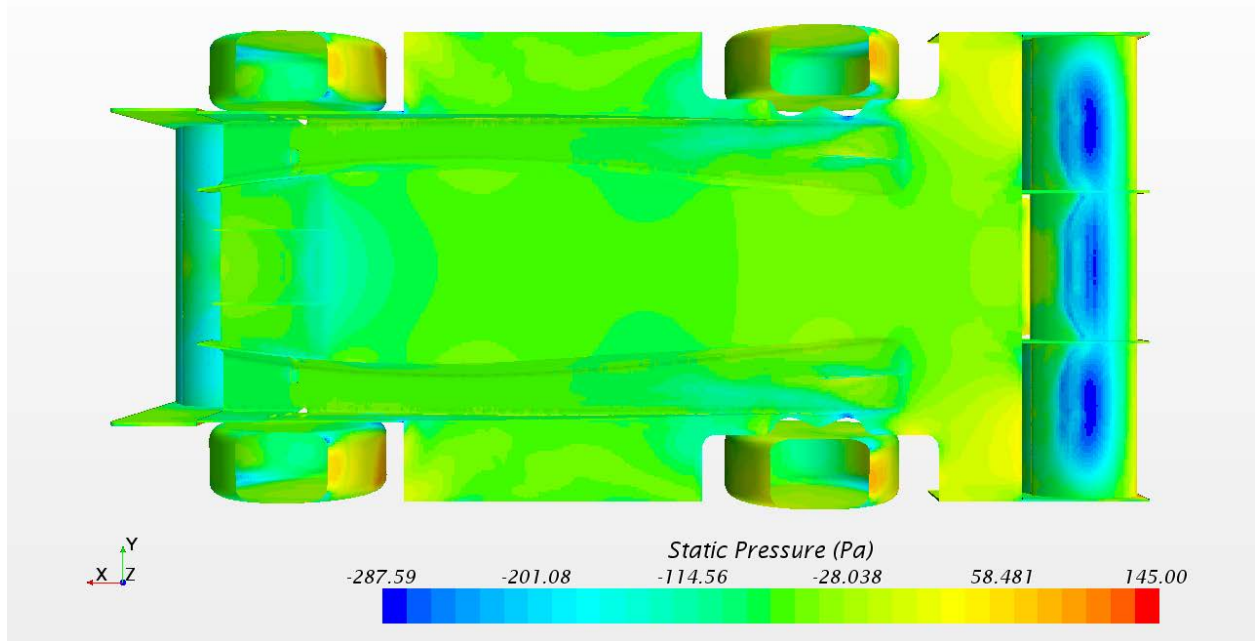


Figure 45: Static pressure scalar underbody

2. Rear wings are working better. We can observe from figure 47 that the flow to the lower wing assembly has improved

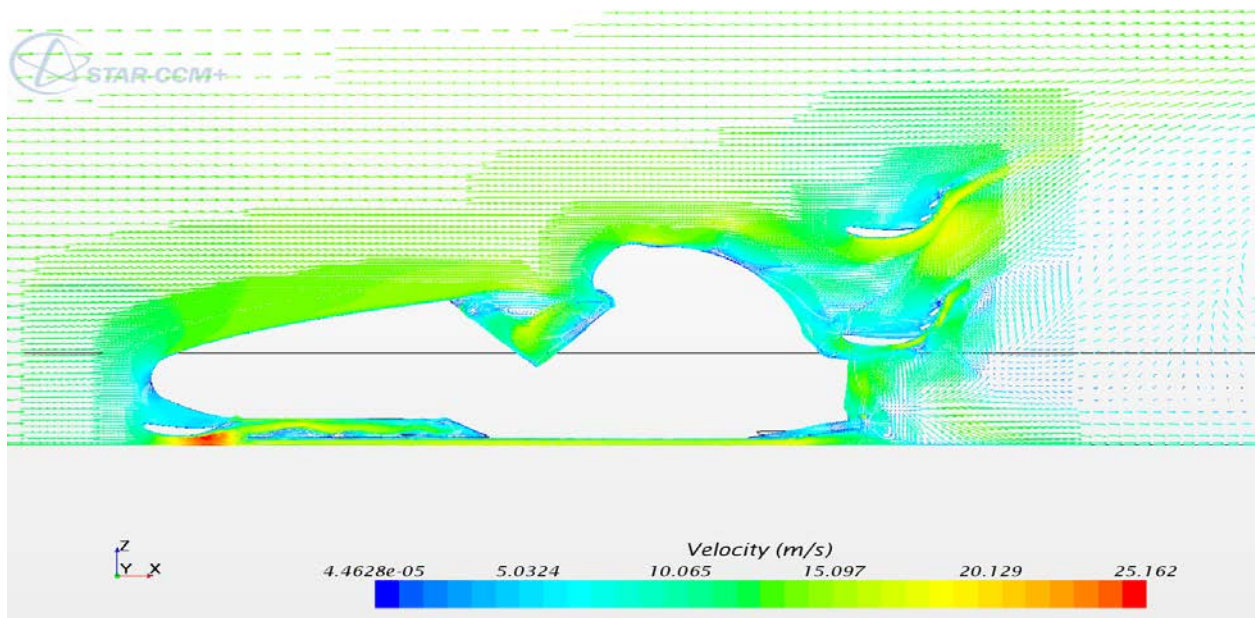


Figure 46: Velocity Vector plot along center section of the car

3. From figure 47 we can also observe that the Diffuser performance has degraded. This is due to the downstream effect of front wing. The front wing creates a wake or turbulence which directs towards the rear and bodywork aerodynamics and has an adverse effect on its performance.
4. There is not enough forward balance. Figure 48 shows the positioning of COP on the car. It is rearwards. Ideally we want the COP to be close to the center of the front and rear axle and slightly rearwards. The front wing can be further rotated another few degrees nose down in order to further increase the front downforce. Also, there is a need to add muffler and radiator duct to CFD model which will increase the front balance and reduces the rear downforce. We can also make an attempt to reduce the rear downforce, we could possibly remove the top flap of the top rear wing making it a 2 element wing to achieve the balance target.

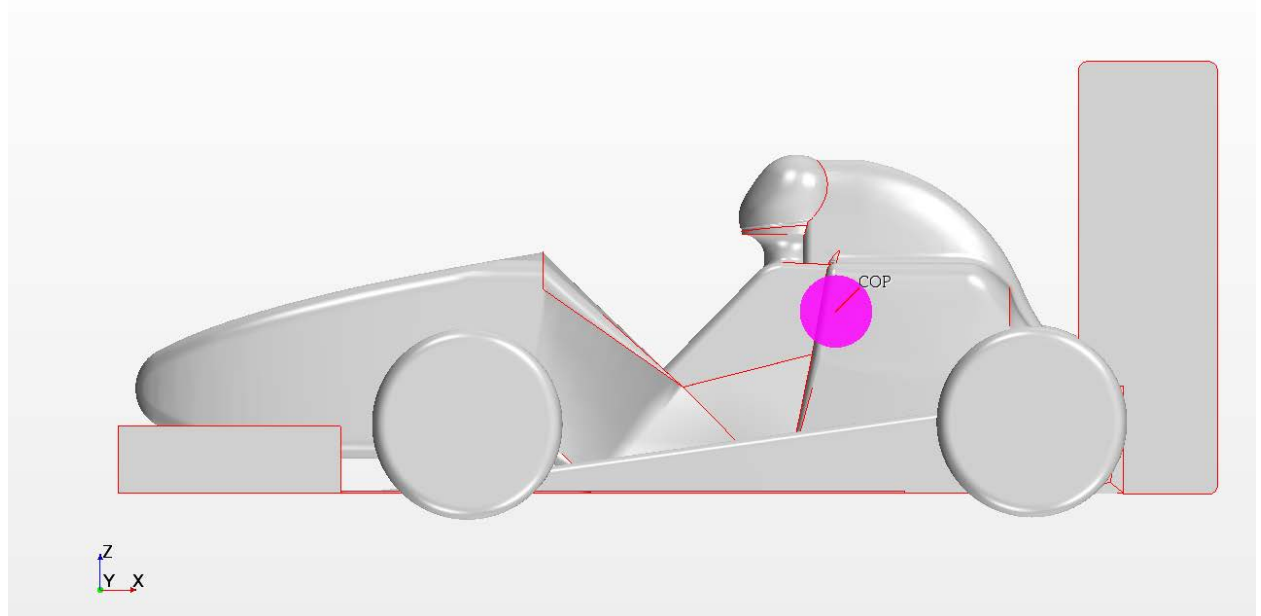


Figure 47: Center of pressure (COP) on the car

5. We can observe from the figure 50 that the leading edge of the diffuser is a square. The flow from under the front wings is separating the flow on top and bottom of the diffuser (Figure 49). Thus there is a need to airfoil shaped diffuser leading edge in order to avoid flow separation. We need to cut the leading edge back away by about 3 inch and then add an airfoil leading edge shape.

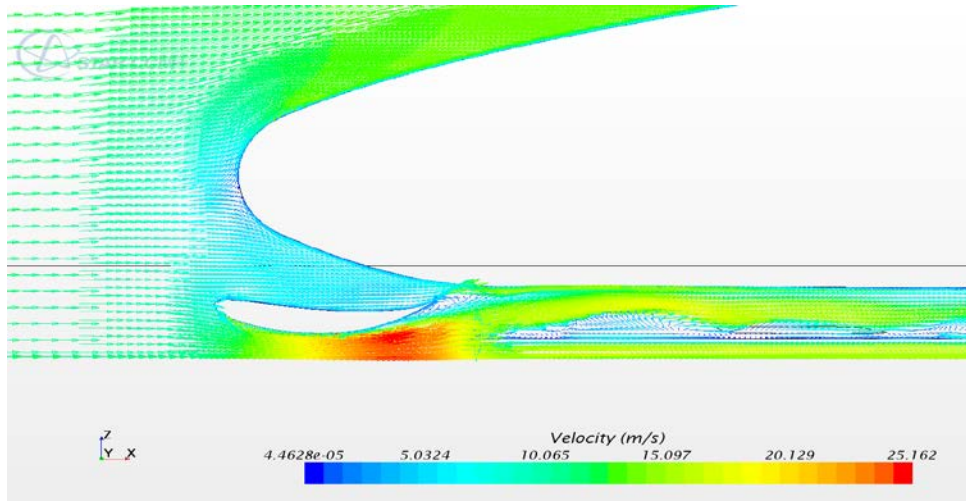


Figure 48: Velocity Vectors showing the flow separation on the diffuser leading edge

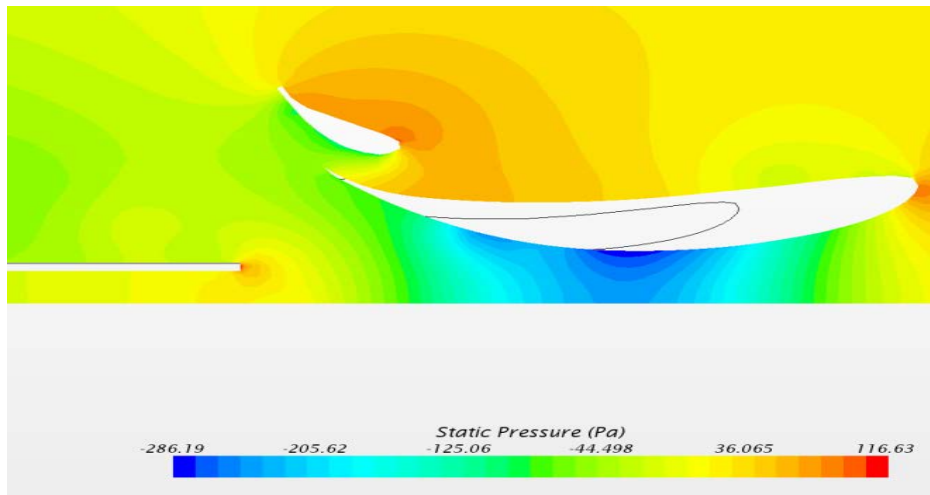


Figure 49: Square edged diffuser leading edge

Design Iteration 5

For this CFD run, the front wing was tested at 7 deg. The leading edge was made an airfoil shape so as to get a good airflow for the underbody. We then added a 1/2" gurney on the trailing edge of the diffuser to close the gap to the lower wing. Top rear wing was changed to two elements. We made the front wing 1" narrower on both sides and add a vertical gurney to the outside of the endplate. The purpose of this was to direct air outside the front tires to help reduce the positive pressure under the floor there. The center part of the wing was made flat in order to get a good airflow to underneath of the car. Figure 51 shows the CAD model for design iteration 5.

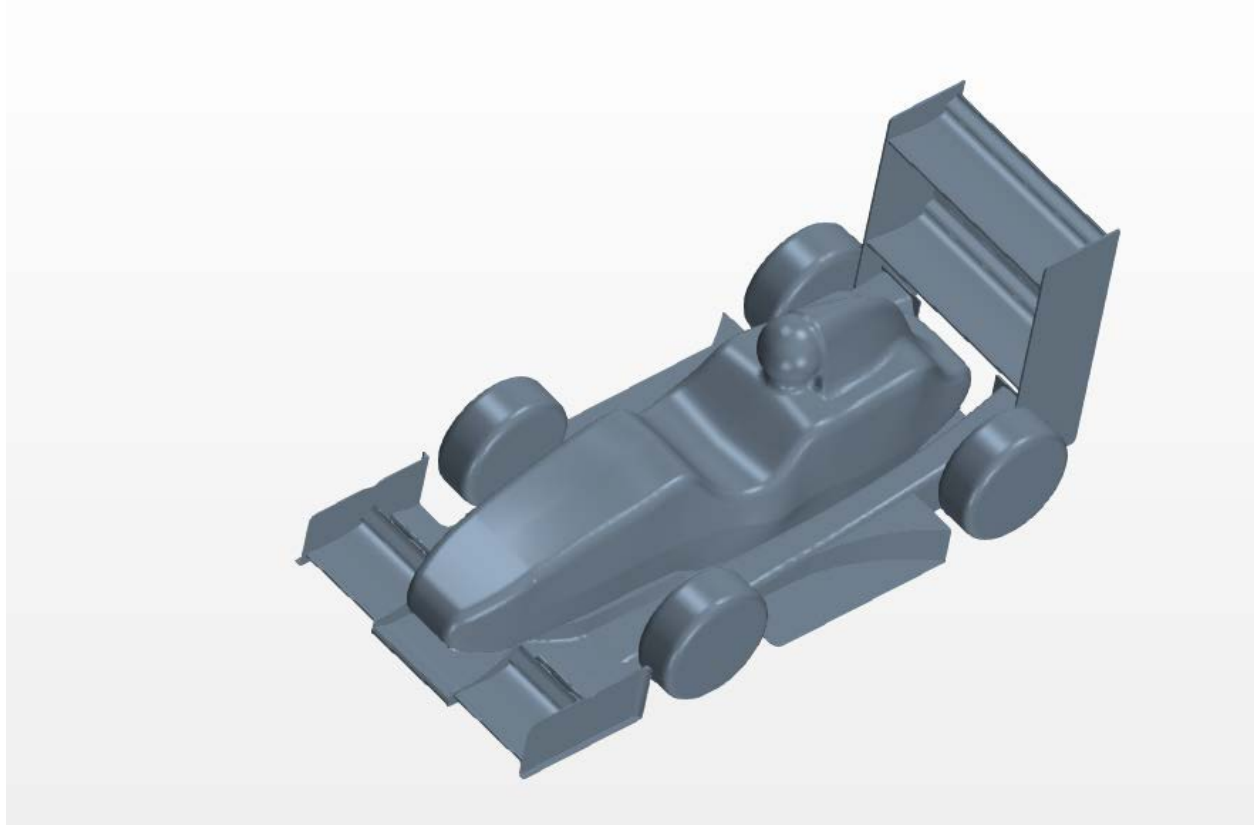


Figure 50: Design iteration 5

The CFD numbers for the above model is as follows:

Parameters	Values
Down force	58 lbs.
Drag	21 lbs.

Frontal area	0.998 m ²
Front Axle balance	47.2

Findings:

1. By having a flat at the center of front wings, the flow to the underbody improved (Figure 52)

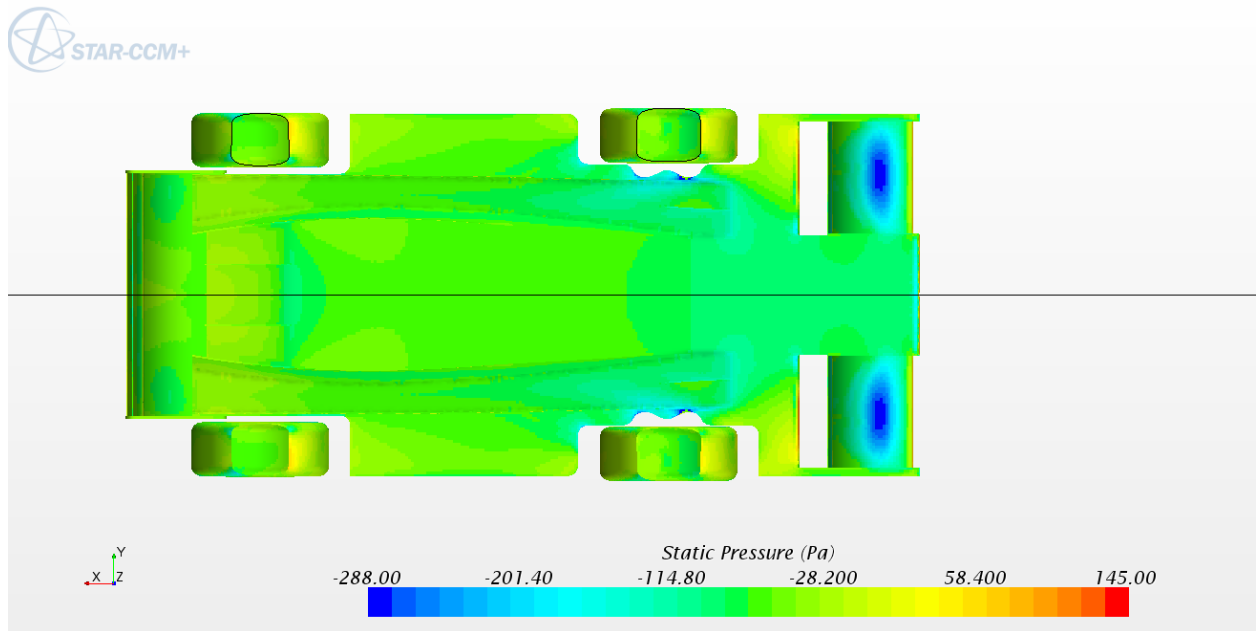


Figure 51: static pressure scalar underbody

2. From the figure 52 we can see that the rear wing performance has decreased. We can barely find low pressure spots on the rear wing which explains the reason for reduced downforce number. Though the balance is improved, but the overall downforce number is decreased. Therefore, there is a need to add more downforce. To improve the downforce number, we should increase the size of aerodynamic devices especially wings.
3. The aero balance has shifted forward to 47.2% front. This is due to two reasons:
 - Loss of performance on rear wings
 - Better airflow to the front part of underbody due to flat inlet.

4. The loss of rear wings performance is due to the upsweep created by the front wings. We can observe from the following image (Figure 53) the high velocity and low pressure wake is produced from the front wing which is heading towards the rear wings. Thus, creating low pressure differential on the top and bottom faces of the rear wings; therefore, reducing the efficiency.

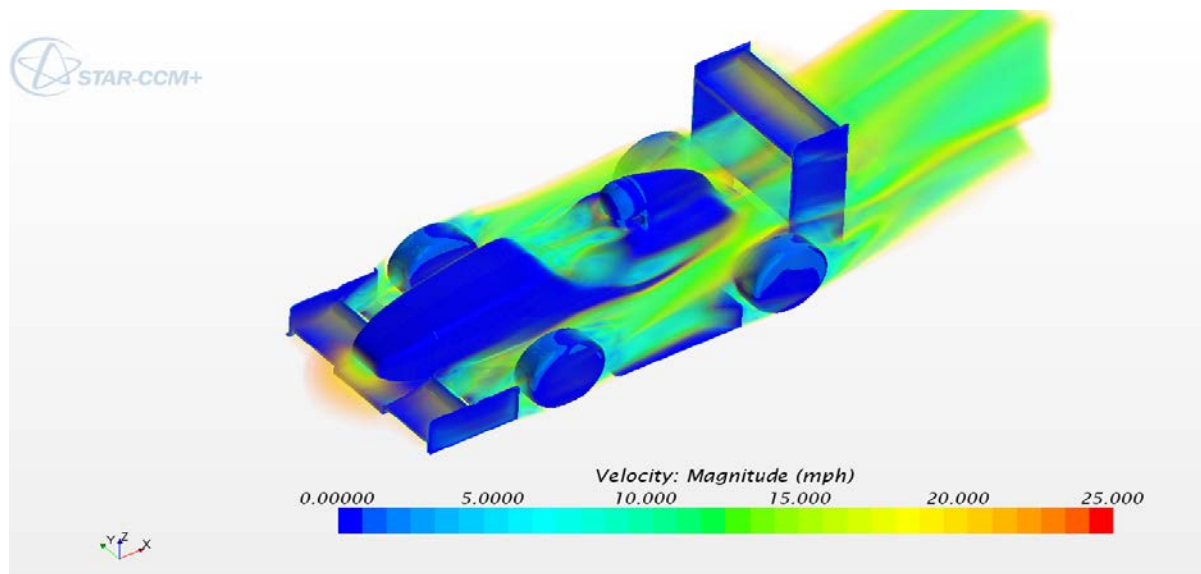


Figure 52: Front wing wake

5. The vertical gurneys in front of the end plates partially manage to push the air outside from striking the front face of the tires (Figure 53).
6. Calculating the center of pressure is not an easy task. To calculate the center of pressure, we had to come with a code which can be run on the STARCCM+ software to yield the result. In order to ensure our code was correct, we did a correlation simulation of this model with the CFD software used in TotalSim. TotalSim LLC is a professional company who are experts in aerodynamics and CFD. We took their help to come up with the CFD

code. We then ran a simulation in both OPENFOAM software and STARCCM+ software.

The results were in good agreement with each other (Except downforce).

Parameters	OPENFOAM	STARCCM+
Downforce	74lbs	58lbs
Drag	21lbs	21lbs
Front balance	47.1	47.2

- Also, we have not yet considered the addition of muffler and the radiator duct. This can have a drastic impact on the rear aerodynamics especially the wake that will be created from the muffler and radiator duct exit can have a massive impact on the rear wing performance; thus reducing rear downforce which means the center of pressure will move further forward.

Design iteration 6

For this model, the muffler and radiator duct were considered. As we know that the radiator exit and muffler will have a huge impact on the rear wings performance, we decided to increase the size of rear wings by 30 % more than wing size used previously so that the overall downforce of the car increases and also have a right aerodynamic balance.

Also we predicted that the radiator duct and muffler would block some airflow heading towards the lower rear wings, we decided to go with 3 elements on top and 2 elements on the bottom. The CAD model is shown below (Figure 54).

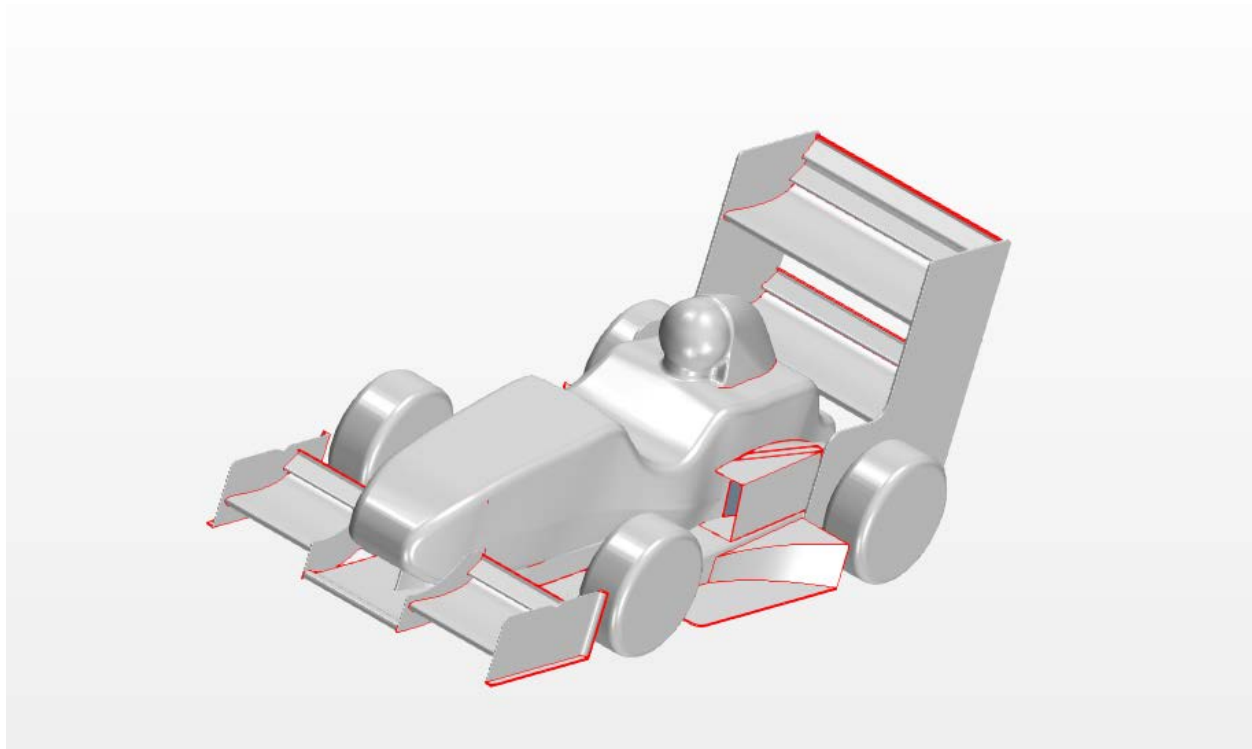


Figure 53: Design iteration 6

The aerodynamic numbers from this CFD results are as follows:

<u>Parameters</u>	<u>Values</u>
Downforce	76.4lbs
Drag	28.7lbs.
Frontal area	1.13 m ²
Cd	1.02
CL	2.73
Efficiency	2.66
Front Axle balance	50.7 %

Observations:

1. Increase in the size of rear wings improves the downforce numbers.
2. The muffler and radiator have an adverse impact on the performance of the lower wing element as we can see some loss of low pressure (blue color) spots on the bottom surface

of the lower wings (Figure 55).

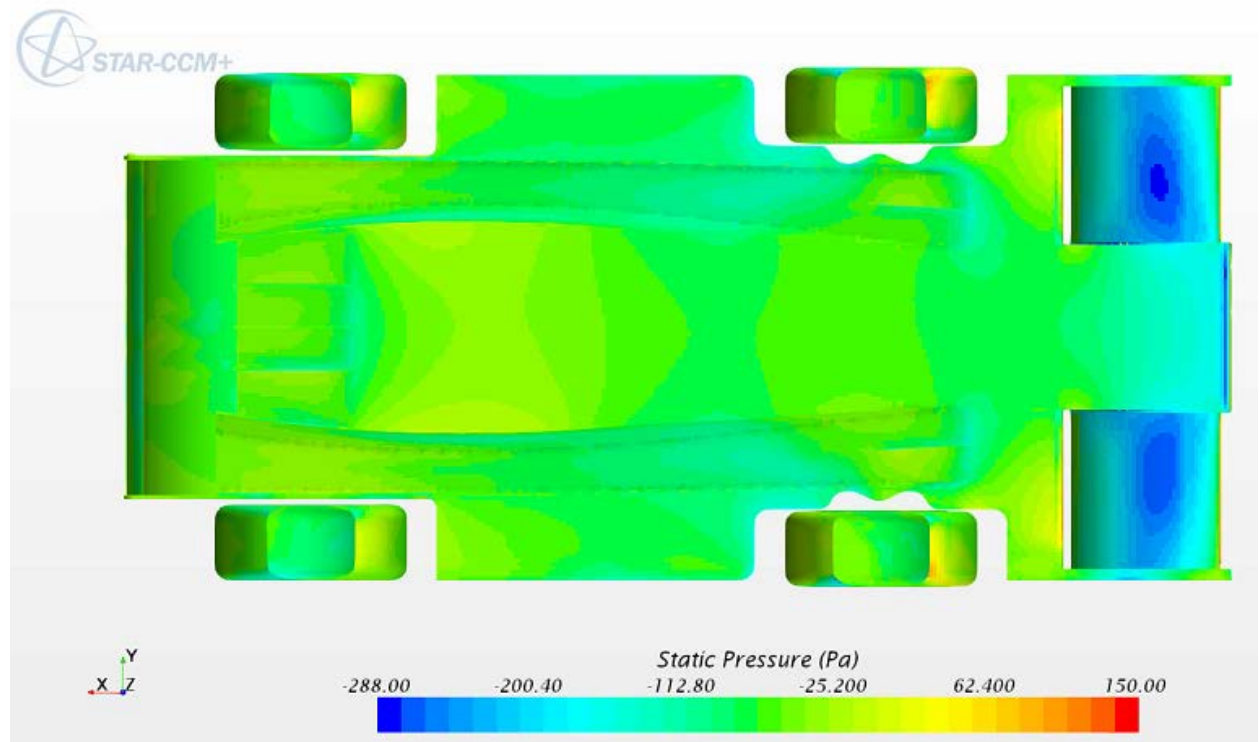


Figure 54: scalar plot: Static pressure plot of the car underbody

3. The aerodynamic balance is better, but ideally we would like to have front balance slightly rearwards i.e. just less than 50 %.
4. There is pressure drop along the radiator. At the radiator duct exit we can see some turbulent or low pressure air. Few streamline were passed through the radiator and we can observe from figure 56 that there is pressure drop and the flow becomes more turbulent. This low pressure and low velocity airflow is heading to the lower rear wing creates a low pressure region on the top surface of wing. Thus, affecting the rear wing performance.

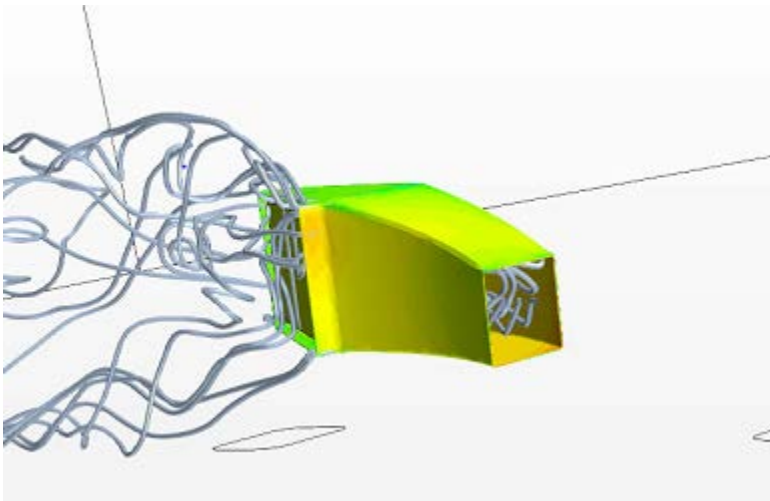


Figure 55: Streamline across the radiator duct depicting the pressure loss

Design iteration 7 (Final CFD model)

The only design change which was done for the final model was changing from 2 element lower rear wing to 3 elements. The purpose of this change was to slightly push the aero balance rearwards and add few extra pound of downforce.

The figure (figure 57 and 58) below shows the final model.

R-CCM+

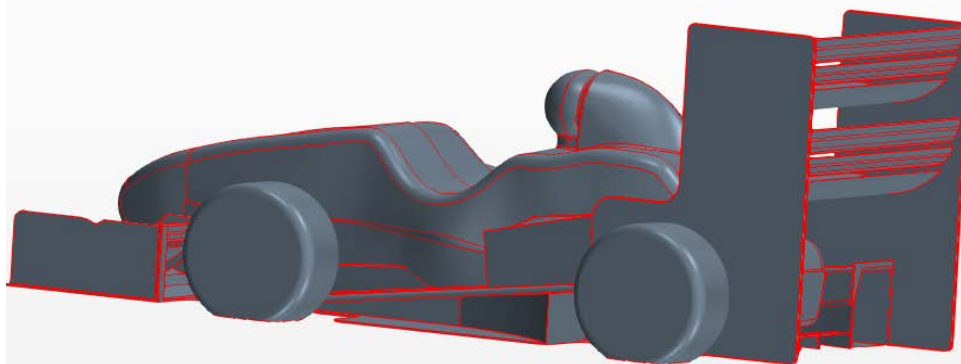


Figure 56: Final design (CFD mode)

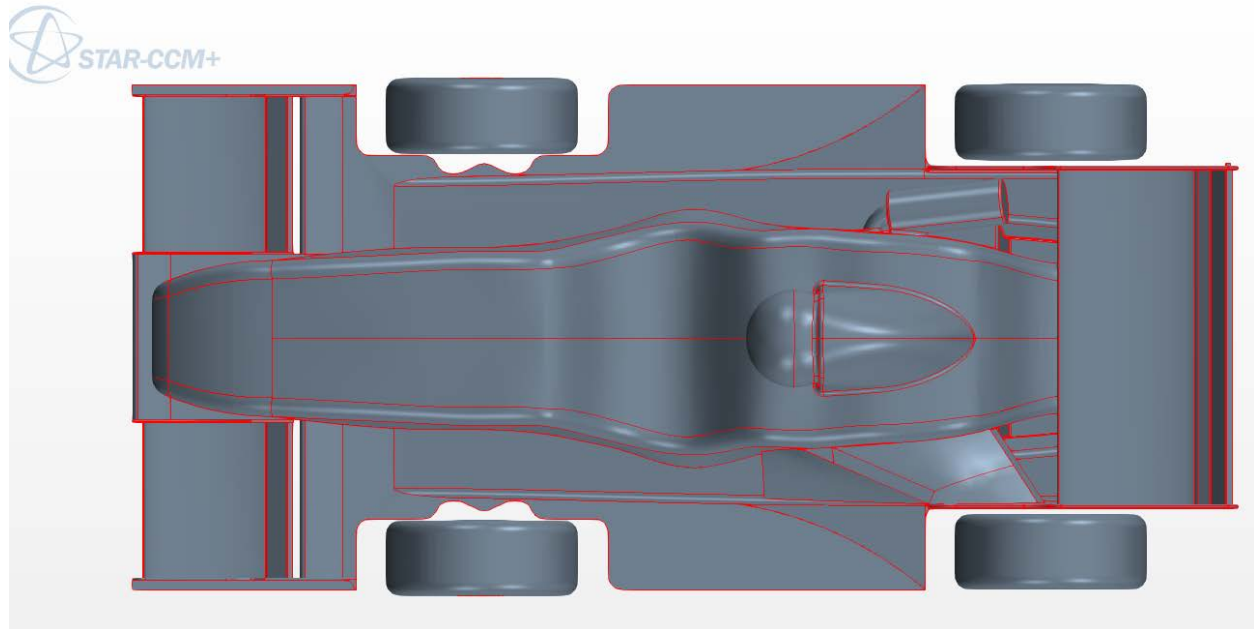


Figure 57: Final Design (CFD model) top view

The CFD results are as follows:

Parameters	Values
Downforce	84.3lbs
Drag	36lbs.
Frontal area	1.23 m ²
Cd	1.17
CL	2.74
Efficiency	2.34
Front Axle balance	48.8 %

Post processing images

The following image shows static pressure plot of the car underbody. We can observe some loss of downforce on the rear wing due the addition of exhaust muffler and radiator which has an adverse effect on the rear wing performance. From the previous design, the bottom rear wings were raised up by 1” in order to improve the performance of rear wings.

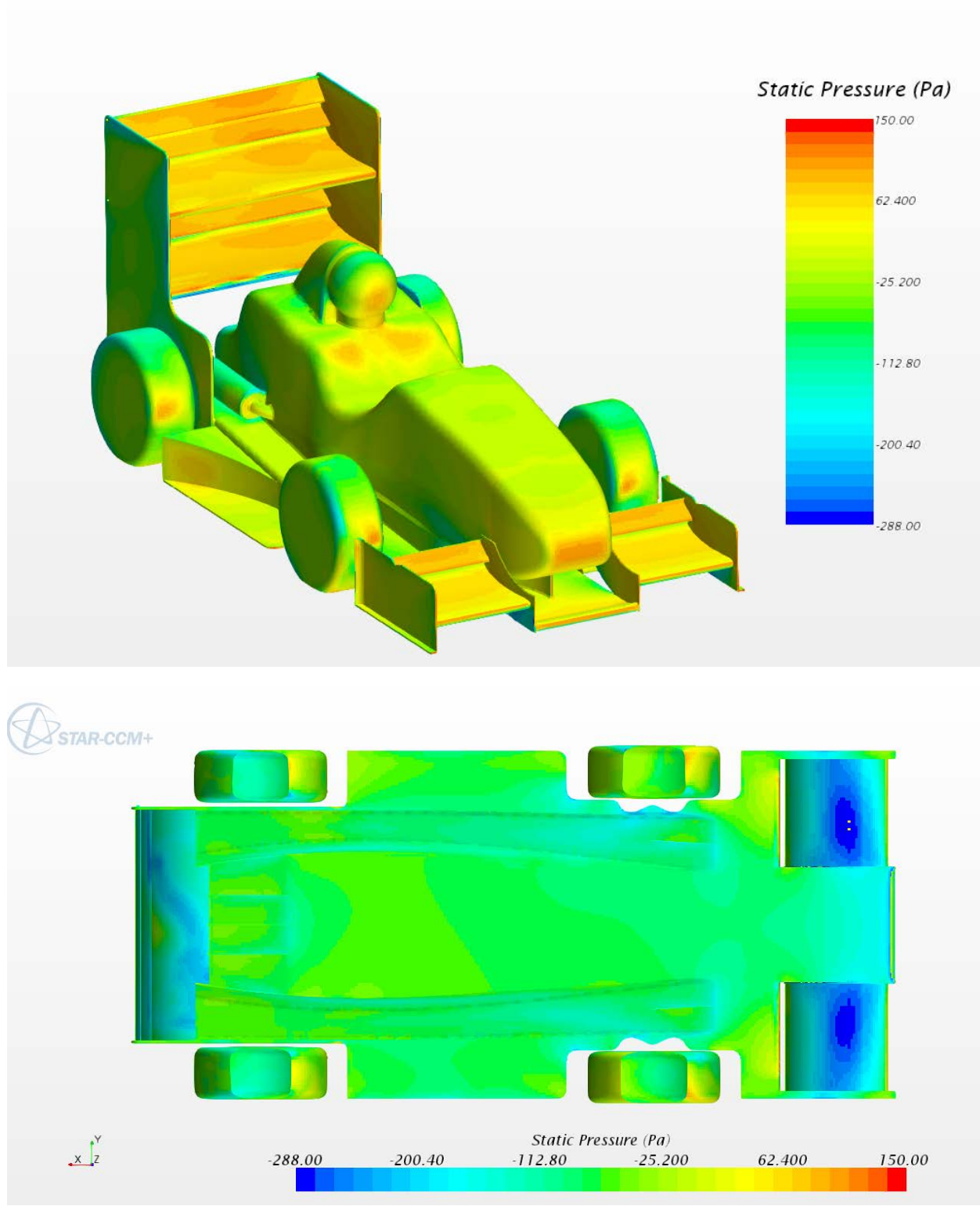


Figure 58: Static pressure plot of car underbody

Figure 52 and 53 shows the velocity vector along the underbody surface and the top surface of the car.

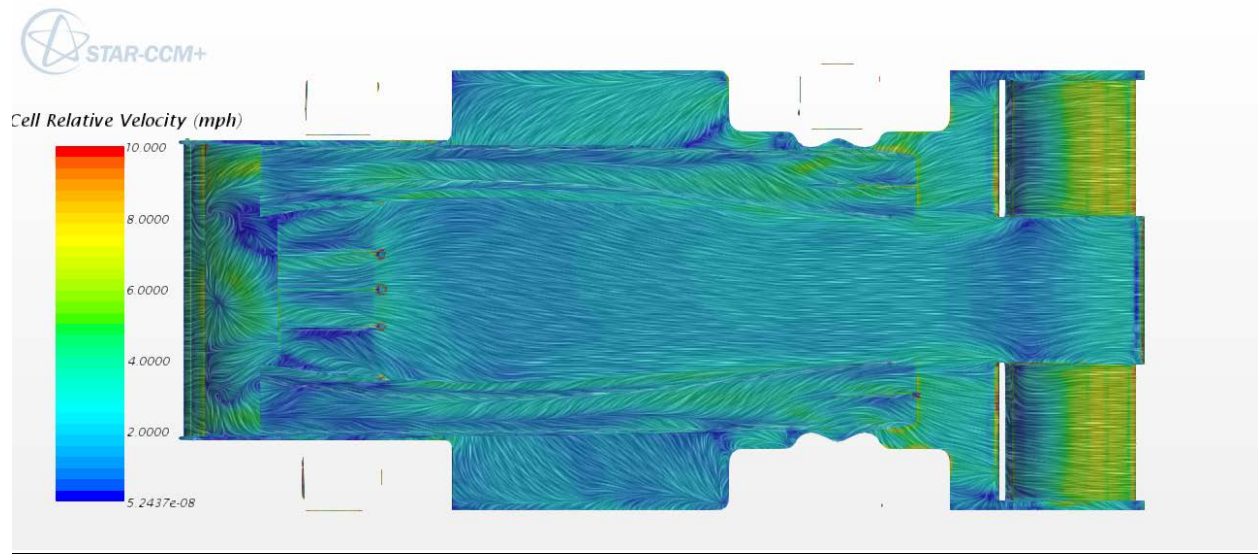


Figure 59: vector plot showing velocity flow along the underbody surface.

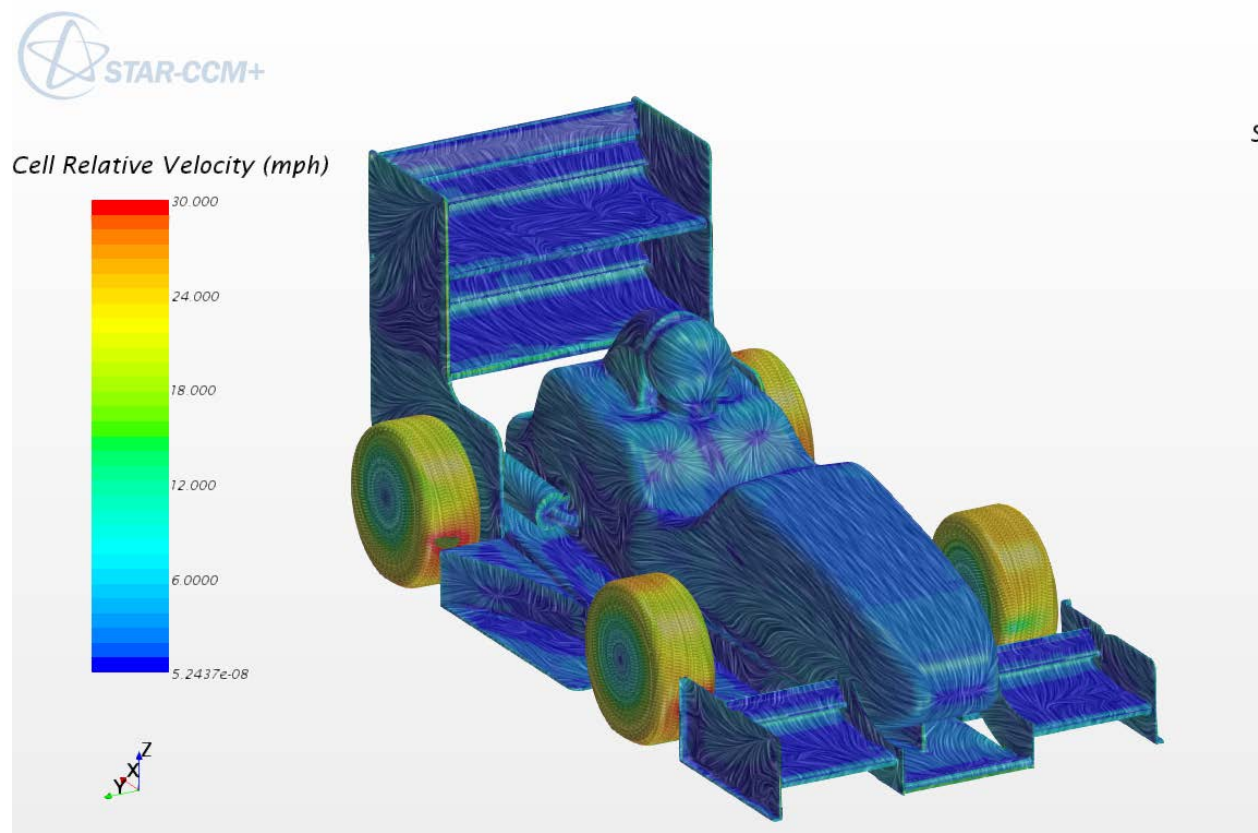


Figure 60: Vector plot showing velocity flow over the top surface of the car.

The following figure (Figure 62) shows the shear stresses along radiator walls. We can observe that the flow is not uniformly distributed indicating poor placement of the radiator. Radiator placement is one area which needs a lot of improvement. Not much study has been done on the radiator duct and its placement on this car. It is something which should be considered in the future.

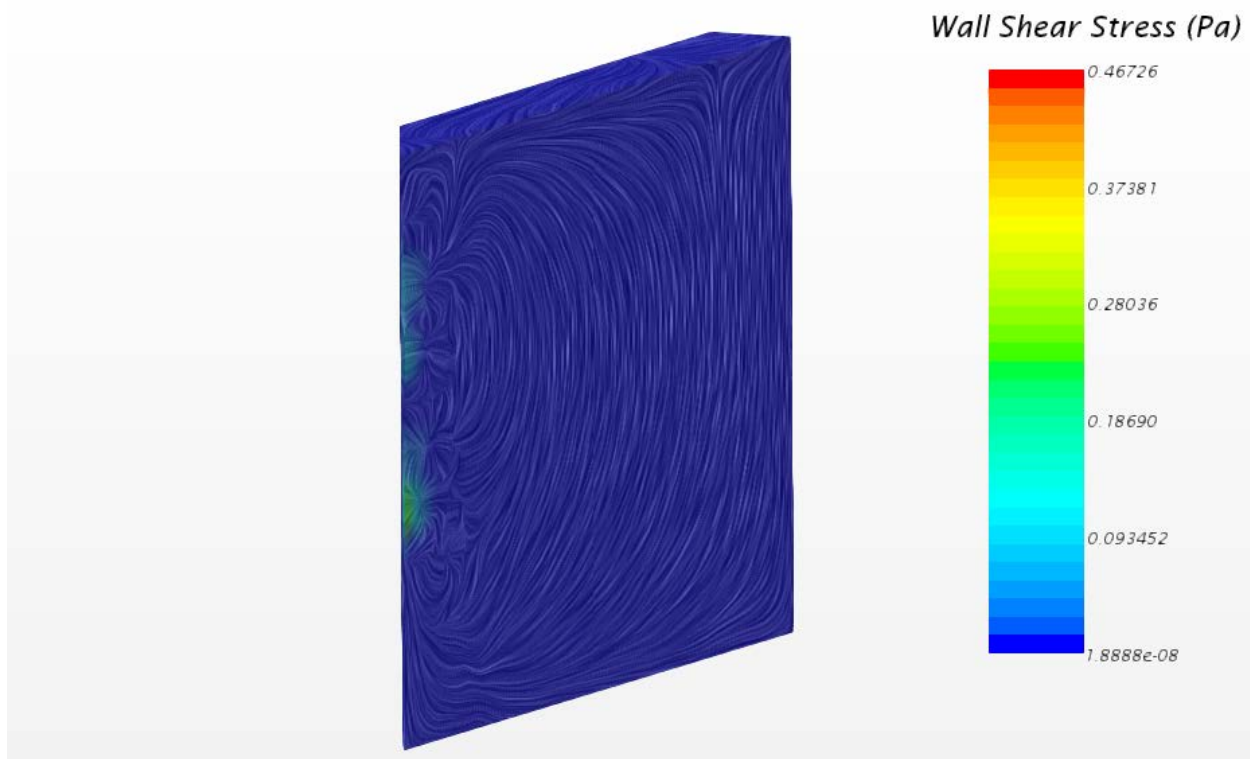


Figure 61: Wall shear plot for the radiator

The figure 63 shows the aerostatic effect (Non Bernoulli effect) of radiator duct exit. We see the blue spots i.e. low velocity behind the duct exit which affects the rear wing performance. It is called non Bernoulli's effect as there is a loss of energy of the airflow when it passes the radiator. There is also pressure drop here. Thus, both velocity and pressure are low at this spot which contradicts Bernoulli's theory. From this observation, we can infer that the best place for

duct is to exit the air into the engine compartment rather than in front of rear wings. This can be considered for the future development.

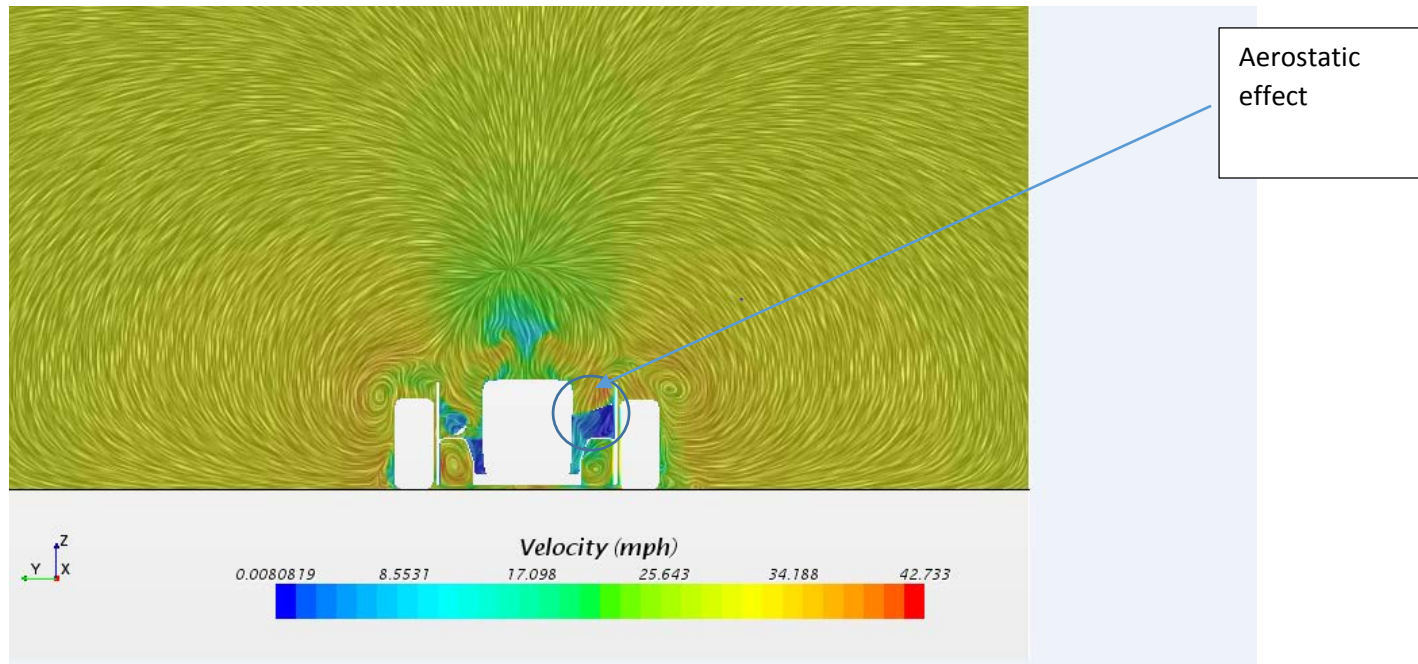


Figure 62: Aerostatic effect of the radiator duct exit

Figure 64 shows the Vortex generation from the end plates. These vortices are in clockwise direction. This is due to the fact that the air moves from high pressure to low pressure. Thus when the Car cuts through the air, wake is produce creating voids just behind the car. These voids are filled by the high pressure air on sides, thus creating a vortex. And the figure 65 shows the rear wing upwash. This process is called vortex shedding. Here the vortex shedding refers to the vortices that are formed continuously by the aerodynamic conditions associated with the car body in the air stream are carried upstream by the flow in the form of vortex street

(<http://dictionary.reference.com/browse/vortex%20shedding>).

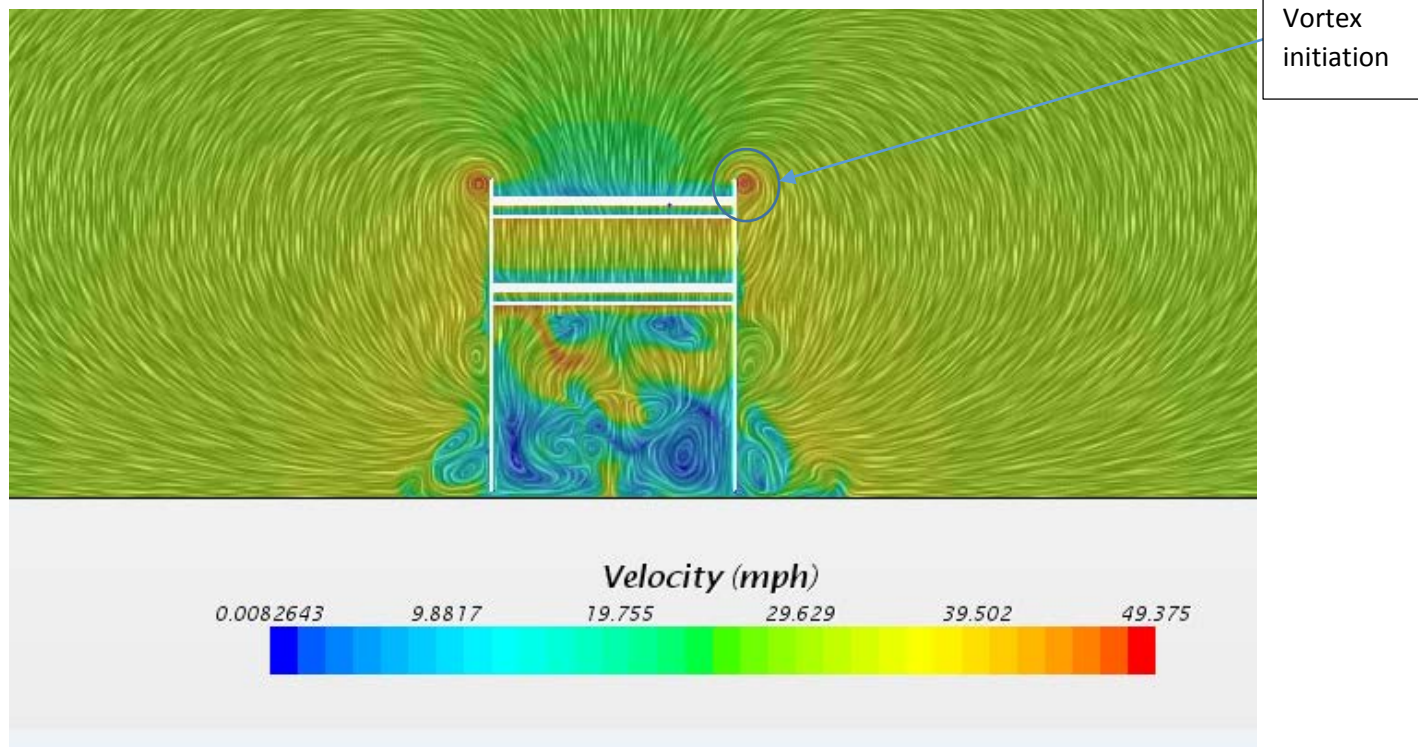


Figure 63: Vortex originating from the rear end plates

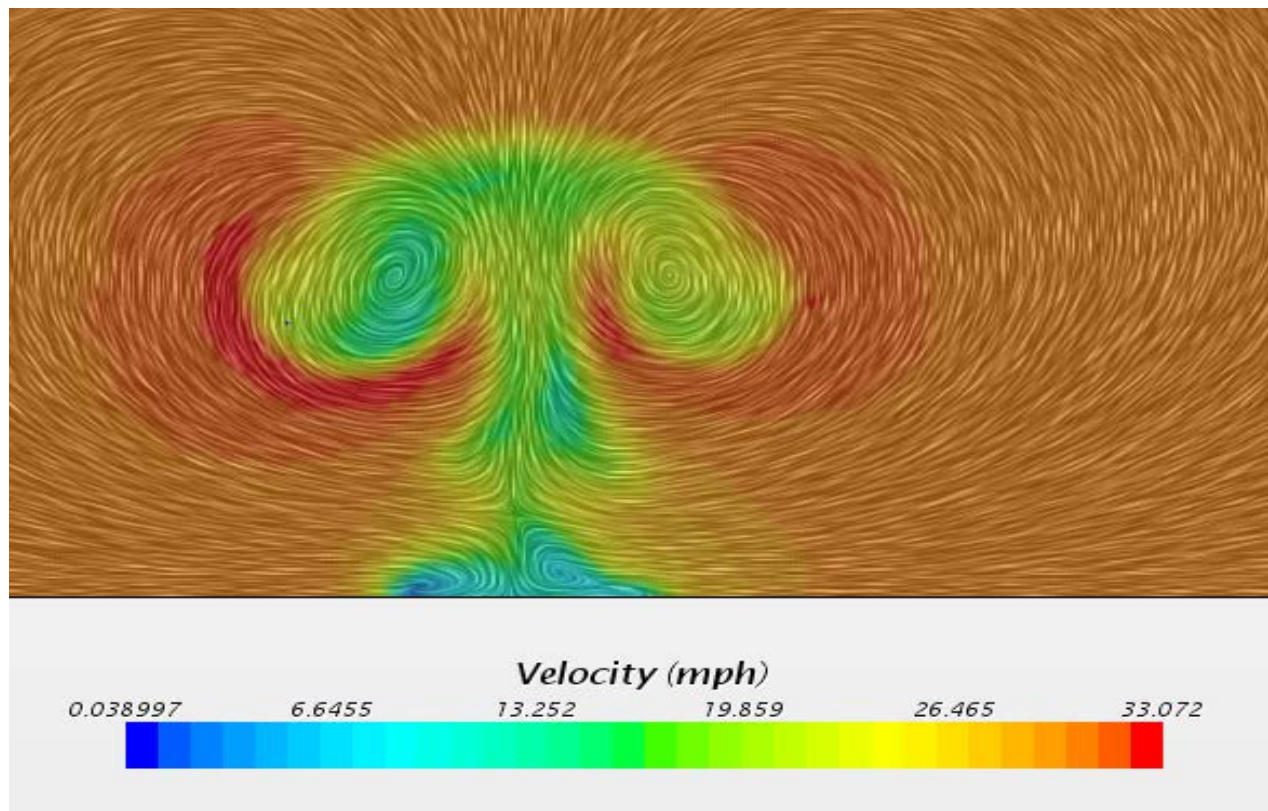


Figure 64: Rear wing upwash

Limitations

- The only major limitation which raised after correlation simulation was the prediction of downforce by OPENFOAM software was not in agreement with STARCCM+ software. OPENFOAM CFD software is used by TotalSim LLC who are a professionals in CFD analysis. There was a difference of about 20 lbs. of downforce. All the other numbers matched perfectly well though.
- As we go on increasing the downforce number, the drag also increases. Increase in drag increases fuel consumption. Thus, we will lose some points in fuel economy part of the competition

Conclusion, Discussion and Recommendation

From this study, the following conclusion can be drawn:

1. Addition of aerodynamic package on the FSAE proves to be an advantage in both static and dynamic part of the FSAE event.
2. CFD simulations on the diffuser ramp angles prove useful data to guide the development of the aerodynamic package for an FSAE car.
3. Wing stall characteristics at low velocity can also be beneficial to develop the wing packages for the car.
4. Diffuser pumping phenomenon enhances the underbody and improves the diffuser performance by creating more low pressure spots on the underbody, thus creating more downforce.

5. Headrest positioning and design has an impact on the rear aerodynamics.
6. Adding an airfoil leading edge to the diffuser on the car enhances the underbody flow.
7. Positioning of radiator duct exit has an adverse impact on the rear aerodynamics.
8. Addition of Gurney Flaps improves the wings performance.
9. The wake arising from the front wings can negatively impact the rear wing performance.

From this study, we have come up with the good aerodynamic package for FSAE car which will be competing in FSAE Michigan 2015. A still further improvement could have been done, but due to the time constraints, the development project has been stopped at this point. The learning from this study will be carried on further for the next FSAE competition which will be competing in 2016. By comparing to the other successful FSAE team, this aero package seems to be good and considering the fact that it's only our 3rd FSAE car and the 1st time we are coming up with optimized aero package, the numbers are quite competitive.

The radiator ducting can have a massive impact on the downforce and drag number, so a further study has to be done in order to design the radiator duct and the radiator placement.

The accuracy of the CFD simulation depends on the mesh size. Thus, more extensive computational resources yields better results. Though CFD is a useful tool for the development process, it is important to realize that CFD is a simulation tool and it just approximates the outside environment. Therefore an on track validation is necessary to investigate the actual performance of car.

References:

- Ehirim, O., "Optimal Diffuser Design for Formula SAE Race Car Using an Innovative Geometry Buildup and CFD Simulation Setup with On-Track Testing Correlation," SAE Technical Paper 2012-01-1169, 2012, doi:10.4271/2012-01-1169.
- Doddegowda, P., Bychkovsky, A., and George, A., "Use of Computational Fluid Dynamics for the Design of Formula SAE Race Car Aerodynamics," SAE Technical Paper 2006-01-0807, 2006, doi: 10.4271/2006-01-0807.
- Wordley, S. and Saunders, J., "Aerodynamics for Formula SAE: A Numerical, Wind Tunnel and On-Track Study," SAE Technical Paper 2006-01-0808, 2006, doi: 10.4271/2006-01-0808.
- Craig, C. and Passmore, M., "Methodology for the Design of an Aerodynamic Package for a Formula SAE Vehicle," SAE Int. J. Passeng. Cars - Mech. Syst. 7(2):575-585, 2014, doi: 10.4271/2014-01-0596.
- Rehnberg, S., Börjesson, L., Svensson, R., and Rice, J., "Race Car Aerodynamics - The Design Process of an Aerodynamic Package for the 2012 Chalmers Formula SAE Car," SAE Technical Paper 2013-01-0797, 2013, doi: 10.4271/2013-01-0797.
- *User Guide STAR-CCM+ Version 8.06.* 2013
- NAFEMS Home engineering analysis and simulation - FEA, Finite Element Analysis, CFD, Computational Fluid Dynamics, and Simulation. (n.d.). Retrieved April 24, 2015, from http://www.nafems.org/.../how_to_understand_cfd_jargon-nafems.p..
- Jawad, B., Longnecker, M., and Timmer, J., "The Impact of Aerodynamics on Vehicle Performance in a Formula SAE Racing Style Vehicle," SAE Technical Paper 2001-01-2744, 2001, doi: 10.4271/2001-01-2744.
- Optimising FSAE Aerodynamics at Monash University. (n.d.). Retrieved from <http://www.computationalfluidynamics.com.au/monash-motorsport-fsae-aerodynamics-guest-post/>
- Tips: Aerodynamics. (1998, January 1). Retrieved from <http://www.gmecca.com/byorc/dtipsaerodynamics.html>

- McBeath, S. (1998). *Competition Car Aerodynamics*. California: Haynes Publishing., ISBN 1 85960 662 8.
- Wong, J. (2008). *Theory of ground vehicles* (4th Ed.). Hoboken, N.J.: Wiley. ISBN 978-0-470-17038-0
- Katz, J. (1995). *Race car aerodynamics: Designing for speed*. Cambridge, MA, USA: R. Bentley. ISBN 978-0-8376-0142-7,
- Smith, C. (1984). *Engineer to win: The essential guide to racing car materials technology*. Rolling Hills Estates, CA: Carroll Smith Consulting., ISBN 978-0-615-75409-3
- *Investigations into vehicle aerodynamics*. (1995). Warrendale, PA: Society of Automotive Engineers., ISBN 1-56091-628-1
- Hucho, W. (1998). *Aerodynamics of road vehicles: From fluid mechanics to vehicle engineering* (4th Ed.). Warrendale, PA: Society of Automotive Engineers, ISBN 0-7680-0029-7.
- http://www.arc.vt.edu/ansys_help/cfx_thry/i1302321.html#i1302649).
- *Emission: Measurement, testing & modeling*. (2006). Warrendale, PA: Society of Automotive Engineers.
- Aird, F. (1997). *Aerodynamics for racing and performance cars*. New York, N.Y.: HP Books, ISBN 1-55788-267-3
- Buckley, F. (1999). ABCD: An improved coast down test and analysis method. *SAE International*, (950626).
- Wordley, S., & Saunders, J. (2006). AERODYNAMICS FOR FORMULA SAE: INITIAL DESIGN AND PERFORMANCE PREDICTION. *SAE International*.
- Brown, M. (2011). *Racecar: Searching for the limit in Formula SAE*. S.I.: Seven Car Publishing., ISBN 0984719318, 9780984719310
- (n.d.). Retrieved April 27, 2015, from [http://encyclopedia2.thefreedictionary.com/computational fluid dynamics](http://encyclopedia2.thefreedictionary.com/computational+fluid+dynamics)
- McBeath, S. (2006). *Competition car aerodynamics*. Sparkford, Yeovil, Somerset, UK: Haynes Pub.

- Clancy, L. (1975). *Aerodynamics*. New York: Wiley.
- E.N. Jacobs, K.E. Ward, & R.M. Pinkerton. NACA Report No. 460, "The characteristics of 78 related airfoil sections from tests in the variable-density wind tunnel". NACA, 1933.
- Ting, L. (1991). *Viscous Vortical Flows*. Lecture notes in physics. Springer-Verlag. ISBN 3-540-53713-9.
- 2.2.2. Two Equation Turbulence Models. (n.d.). Retrieved April 27, 2015, from http://www.arc.vt.edu/ansys_help/cfx_thry/i1302321.html#i1302649.
- (n.d.). Retrieved April 27, 2015, from https://steve.cd-adapco.com/articles/en_US/FAQ/SW-5-262
- *2015 FORMULA SAE RULES*. (2014). SAE INTERNATIONAL).
- Giguere, P.; Lemay, J.; Dumas, G. (1995). "Gurney flap effects and scaling for low-speed airfoils". *AIAA Applied Aerodynamics Conference, 13 th, San Diego, CA, Technical Papers*