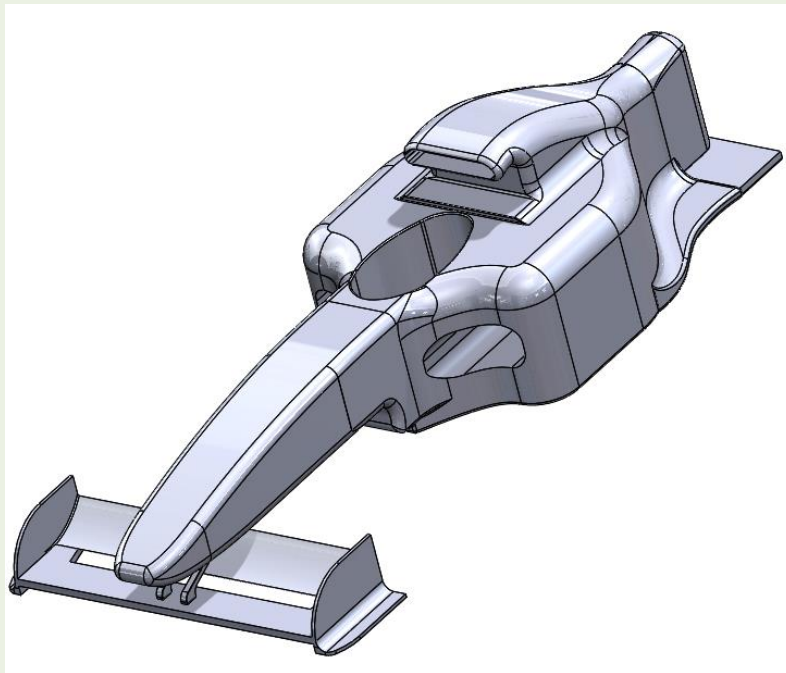




**UTS University of Technology Sydney**

**Faculty of Engineering and IT**

**Advanced Flow Modelling**



**AMAN SANGRAM SINGH**

## CONTENT

NO	NAME
1	Approaching a CFD problem
2	Solving using ANSYS workbench
3	STEP 1: Creating Geometry
4	STEP 2: Meshing
5	STEP 3: Analysis
6	Solution/Results
7	Converging grid graphs
8	Velocity Contours
9	Pressure Contours
10	Velocity Streamline
11	Velocity Vector
12	Drag and Lift coefficient
13	Discussion and Conclusion

# Advanced Flow Modelling

## CFD Individual assignment

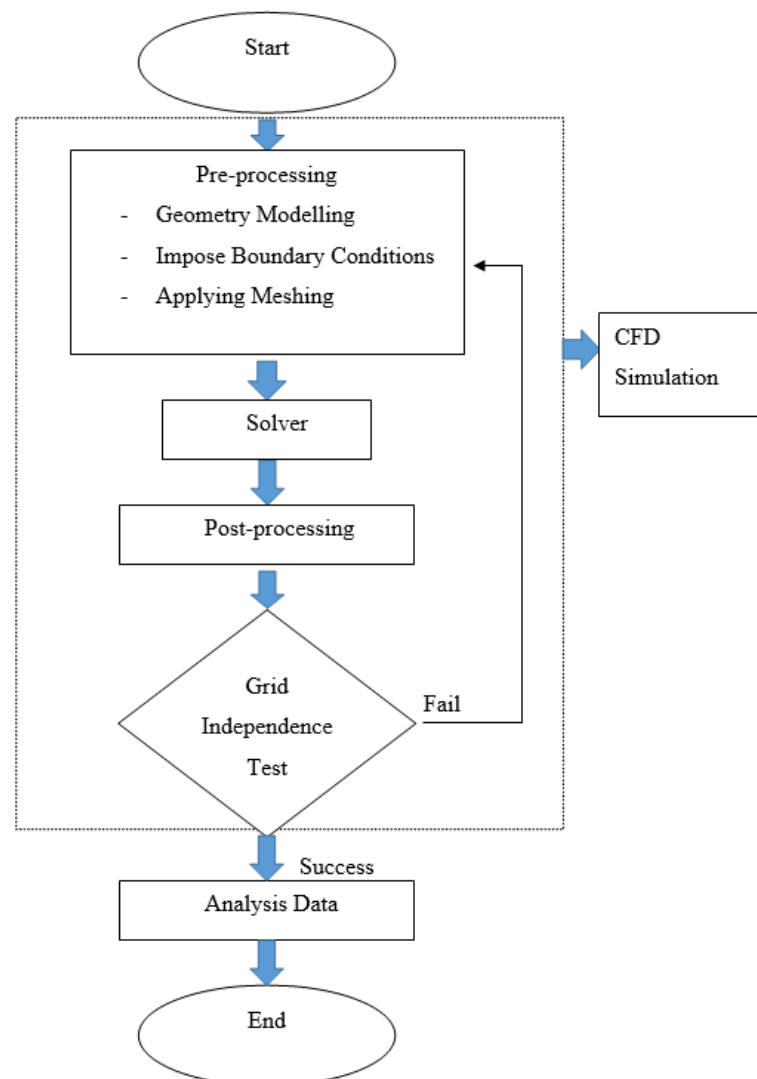
Problem: Analysis of Formula-1 car

Procedure and discussion points:

1. Create the computational geometry in DesignModeler/ Solidworks/any CAD software with the mentioned dimensions.
2. Build an appropriate mesh for the model and **perform a grid refinement study**.
3. Run the simulation for different flow condition (At least two different inlet condition. More than 2 are encouraged).
4. Using the converged grid, **check the iterative convergence**.
5. **Draw the Velocity contour** at different selected planes of your model and **discuss your results**.
6. **Draw the pressure contour** at different selected planes of your model and **discuss your results**.
7. **Plot the temperature profile** along a selected line from inlet to outlet and **discuss**.
8. **Draw the wall shear stress** and **discuss**.
9. **Draw the turbulence intensity** contour for different flow rates and discuss (If applicable)
10. **Plot the axial velocity profile** at any selected position of your model and **discuss**.
11. **Draw the Velocity vectors** at different selected planes of your model and **discuss your results**.
12. **Calculate** the drag and lift coefficient and **discuss (if applicable)**.
13. **Calculate** the deposition efficiency/deposition fraction and **discuss (if applicable)**.
14. **Draw the particle trajectories** for different flow rates and **discuss (if applicable)**.
15. **Calculate** the heat transfer coefficients **(if applicable)**.
16. **Calculate** the Nusselt number from the following correlation and discuss **(if applicable)**;  
$$Nu_D = 0.023 Re_D^{4/5} Pr^n$$
17. Connect your results with the existing literature.

## 1. Approaching a CFD problem

The figure below shows main steps/stages for proceeding from start of the problem to the solution. Starting off with the basic steps which involves making a geometry model using a CAD software, applying boundary condition to it and then creating the mesh. This part is known as pre-processing as this involves all the requirements which are needed to be done before the analysis starts. Feeding the model to the solver will use various mathematical models to solve it. Steps after this are called post-processing which mainly involve checking the results is the simulation was successful. We can get various solutions depending upon our needs, different graph plots, contours for velocity and pressure, stress strain graphs etc

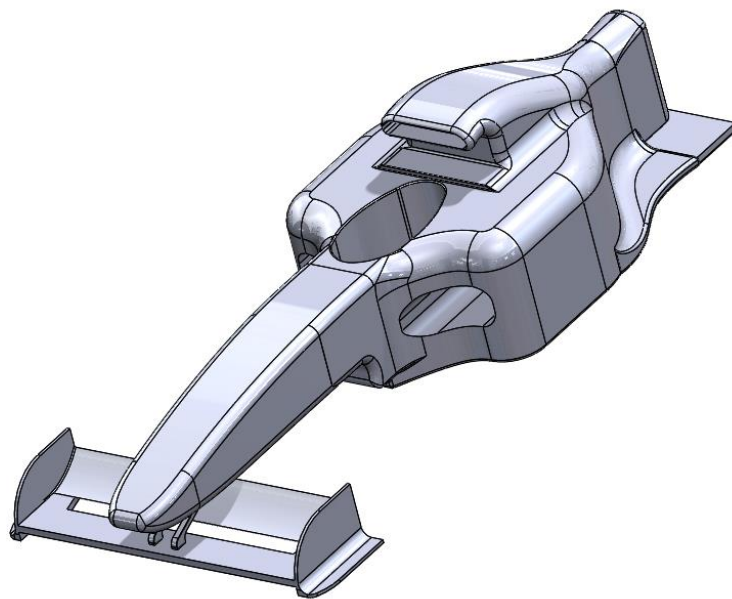


## 2. Solving using ANSYS workbench

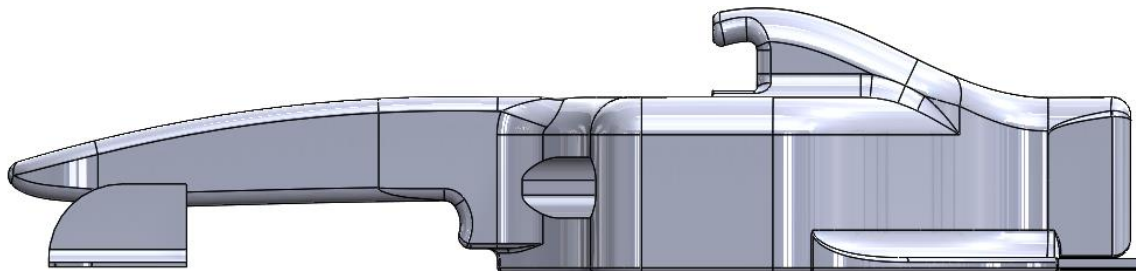
Below are the steps that were performed to solve the given air flow problem using the ANSYS workbench

### 2.1 STEP 1: Creating Geometry

The first step for starting off with the CFD analysis involves creating a geometry or a 3D CAD model of the structure that we are trying to solve. For the given problem solidworks was used to create a model of an F1 car. The model includes the front wing, nose, cockpit, side air vents and a bit of back body. Tires were excluded from the model as they have a standard shape and size, which cannot be modified. Also, this made the model much computationally feasible.



For the geometry of the car, first of all a 2D sketch for half base outline was created with desired dimensions and then extrude-boss function was used to make a solid body or into a 3D model. Further fillet functions were used to define the curvature of various sides and edges of the car. Then mirror function was used to make a whole body from half geometry. This reduced the time required for making that whole body by half. The basic structure or body of the car was ready. After which side air vents and the cockpit were made using the extrude-cut function. Next was the nose of the car which will further hold the front wing.



The total length of the F1 car model is 595mm from front wing to the back plate. It is 200mm wide and stands at a height of 140mm from base to air intake.

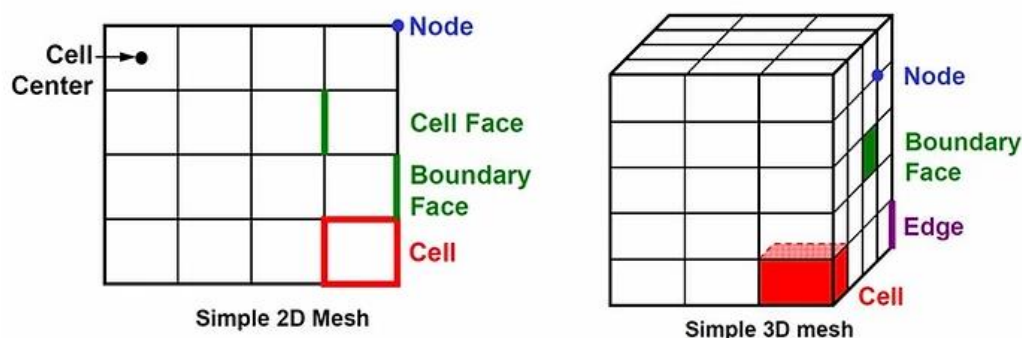
After the model was created in Solidworks, it was imported into ANSYS workbench and there it was edited in design modular. In DM a bounding box was created around the car which would define the space for fluid flow. Final step involved naming the sides of the bounding box. Naming the inlet, outlet and the walls will further help in simulation

## 2.2 STEP 2: Meshing

Next step in the process is meshing. Creating a mesh from the given model, which is an essential step of this whole process. It directly effects the computational feasibility i.e. time required and the accuracy of the result. A poor-quality mesh often ends up with a poor-quality solution. But a very high mesh count not always guarantee a highly accurate solution.

The main purpose of mesh generation is to divide the model or the domain into sub-domains or smaller parts, known as cells or elements. Some terminology related to mesh generation is as follows:

Cell	Control volume into which domain is broken up
Node	Grid point
Cell Centre	Centre of a cell
Edge	Boundary of a face
Face	Boundary of a cell
Zone	Grouping of nodes, faces & cells
Domain	Group of node, face & zone



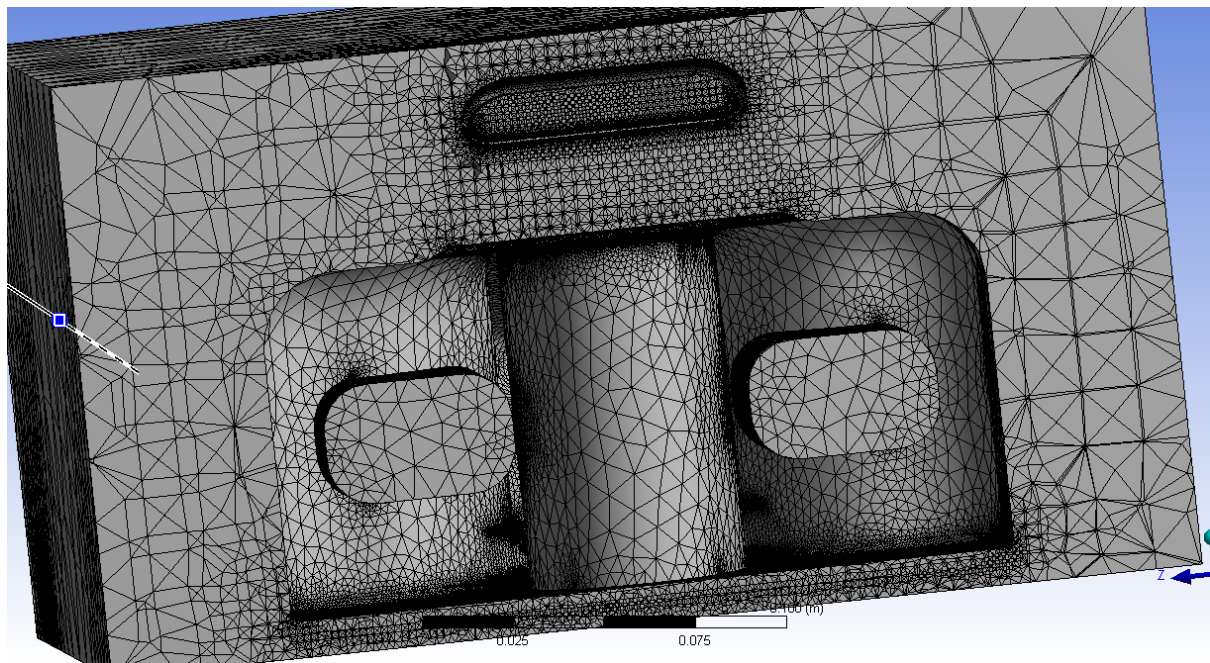
The first step for meshing the CAD model was first creating the bounding box around it. This was done in design modular. A rectangular box was created around the geometry and then naming was done as,

**Inlet-** It is the face of the cuboid that is in front of the nose of the car.

**Outlet-** It is the face which is at the back of the car and is the place where the air will exit.

**Wall-** This refers to the four sides of the box which surrounds the geometry, air will flow within this and this is the main volume for simulation.

Patch independent method was used to generate the mesh for this model. This the same method, used in ANSYS ICEM for mesh generation. For CFD analysis this is a good option to go for. Here starting from the side, a tetra-mesh is generated. It helps in avoiding the point in the geometry which have no impact on our solution or relevance to our geometry.



## Mesh parameters

For meshing the following parameters were used:

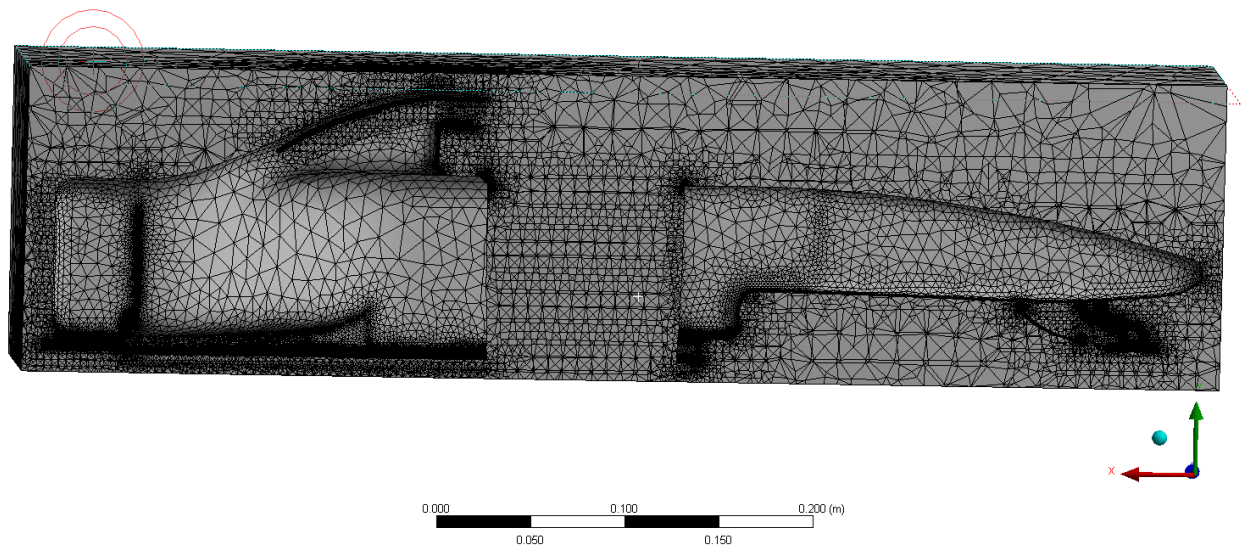
Object Name	Mesh
State	Solved
<b>Display</b>	
Display Style	Use Geometry Setting
<b>Defaults</b>	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	Default (3.62e-002 m)
Export Format	Standard
Export Preview Surface Mesh	No
<b>Sizing</b>	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	5.43e-002 m
Mesh Defeaturing	Yes
Defeature Size	Default (1.81e-004 m)
Capture Curvature	Yes
Curvature Min Size	2.54e-004 m
Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	0.724 m
Average Surface Area	2.3092e-003 m²
Minimum Edge Length	2.9748e-006 m
<b>Quality</b>	
Check Mesh Quality	Yes, Errors
Target Skewness	0.4
Smoothing	High
Mesh Metric	Orthogonal Quality
Min	4.1998e-002
Max	0.99085
Average	0.71232
Standard Deviation	0.16706

Object Name	Geometry
State	Fully Defined
<b>Definition</b>	
Source	C:\Users\13681085\Downloads\final_ass_files\dp0\FFF\DM\FFF.agdb
Type	DesignModeler
Length Unit	Meters
<b>Bounding Box</b>	
Length X	0.63745 m
Length Y	0.16344 m
Length Z	0.30185 m
<b>Properties</b>	
Volume	2.6967e-002 m³
Scale Factor Value	1.
<b>Statistics</b>	
Bodies	1
Active Bodies	1
Nodes	943230
Elements	4843046
Mesh Metric	Orthogonal Quality
Min	0.041997943937863
Max	0.990854150915676
Average	0.71232106853154
Standard Deviation	0.167064383496125
<b>Update Options</b>	
Assign Default Material	No
<b>Basic Geometry Options</b>	
Parameters	Independent
Parameter Key	
Attributes	Yes
Attribute Key	
Named Selections	Yes
Named Selection Key	
Material Properties	Yes

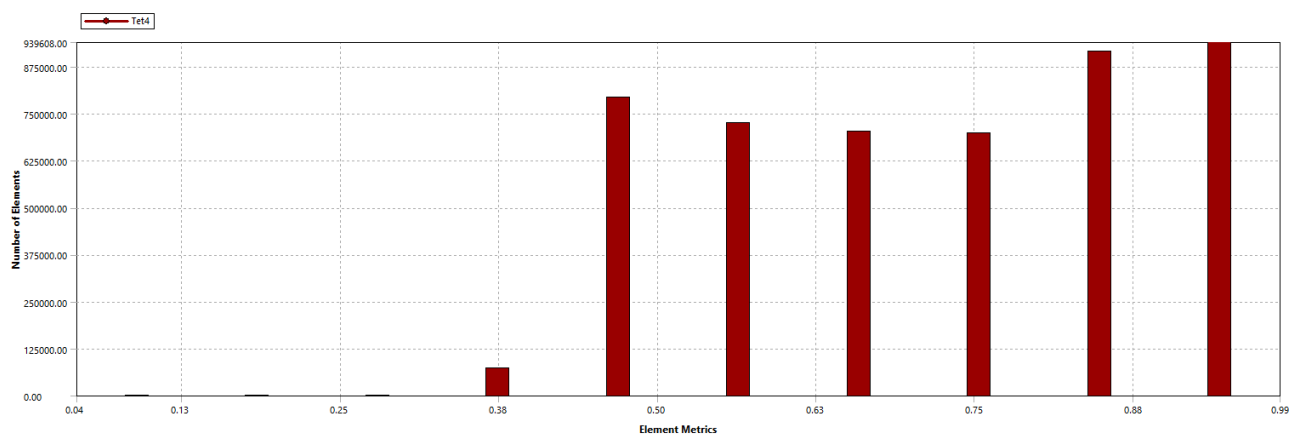


## Grid Refinement

Near the crack edge, mesh refinement is added. Typically, features of quarter-point individuality are Using the. By moving the mid-side nodes to the quarter position at the quarter position, these elements are obtained. Crack tip for reaching the singularity of theoretical square root tension for cracks inhomogeneous Yeah. Medium. Near the crack tip, the final mesh should be fine enough to capture the high-stress Shift. It is of profound significance to measure and minimise the error due to discretization in Analysis of numbers.



Controls



### 2.3 STEP 3: Analysis

#### Expressions for Drag coefficient and Lift coefficient

An important part of air flow analysis is, determining the drag coefficient and lift coefficient. For the F1 car, above mentioned coefficients were obtained using the following expressions:

##### Drag Coefficient:

$$2 * \text{force\_x}()@ \text{Airfoil} / (\text{area}()@ \text{Airfoil} * \text{massFlowAve}(\text{Density})@ \text{Inlet} * (\text{massFlowAve}(\text{Velocity})@ \text{Inlet})^2)$$

##### Lift Coefficient:

$$2 * \text{force\_y}()@ \text{Airfoil} / (\text{area}()@ \text{Airfoil} * \text{massFlowAve}(\text{Density})@ \text{Inlet} * (\text{massFlowAve}(\text{Velocity})@ \text{Inlet})^2)$$

Here,

$$C_d = 2F_x / (\rho A v^2)$$

$$C_L = 2F_y / (\rho A v^2)$$

$C_d$  = drag coefficient

$C_L$  = lift coefficient

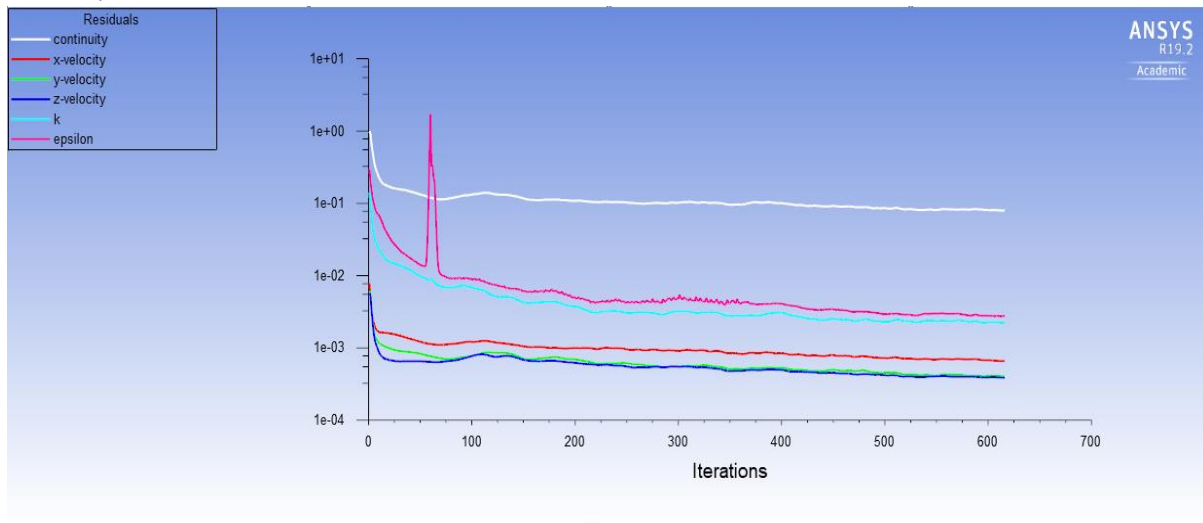
$F_x$  = Force along X-direction

$F_y$  = Force along Y-direction

### 3. Solution/Results

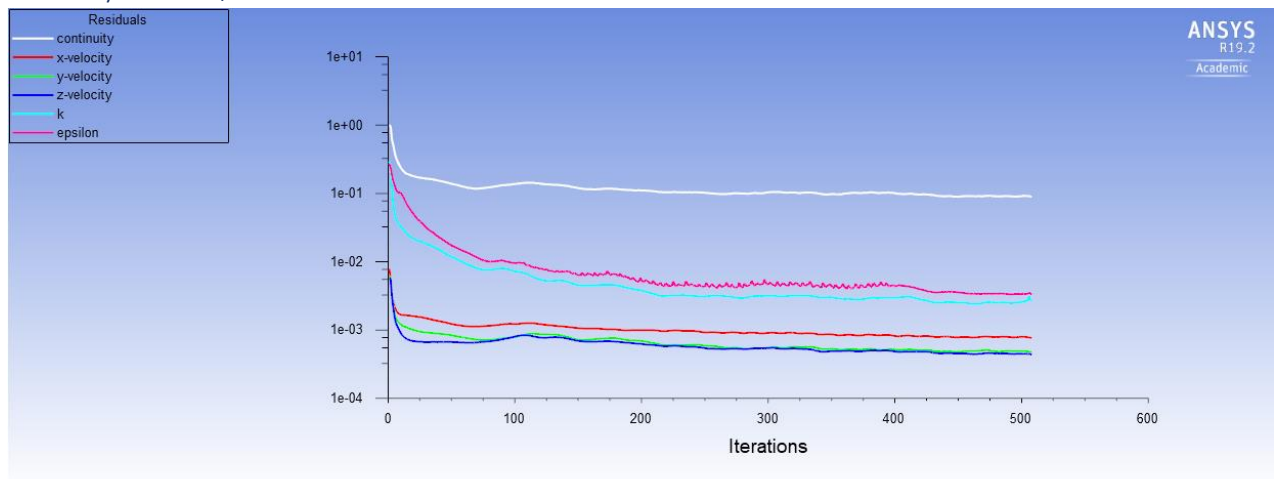
#### 3.1 Converging grid graphs

Velocity= 50 m/s



The following graph represents the x, y and z velocity, along with continuity, k and epsilon. In this case where we are taking velocity to be 50m/s, we can see that the X velocity , Y velocity and Z velocity almost reached the value somewhere between 0.001 and 0.0001 after 650 iterations. While k epsilon reached a value halfway between 0.01 and 0.001 after same number of iterations. After 650 iterations continuity reached only 0.1

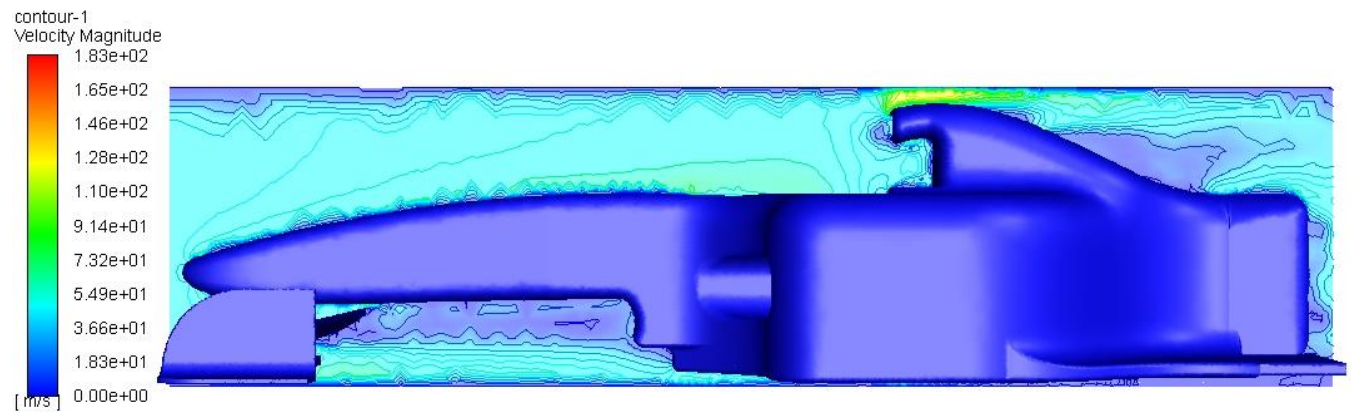
Velocity= 150 m/s



When we took velocity of 150m/s the graph showed that continuity reached 0.1 after about 500 iteration but still struggled to converge. X, Y and Z velocities reached below 0.001 and k and epsilon reached a value of below 0.01 after 500 iterations.

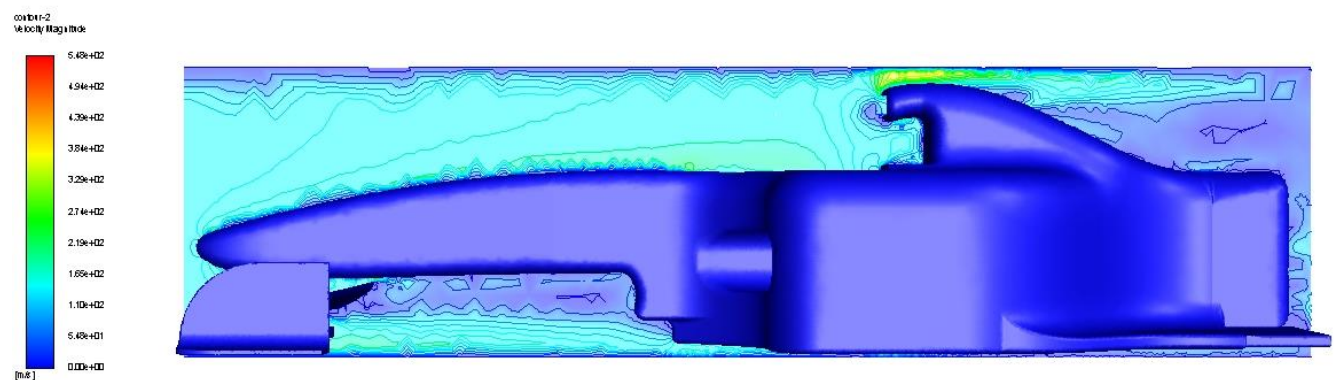
### 3.2 Velocity Contours

Velocity = 50 m/s



In the velocity contour where velocity was 50 m/s we can see that the velocity started off at 50, but after hitting the nose and front wing of the car it starts increasing a bit and reaches to about 80m/s when it reaches near the cockpit. The velocity is maximum at the top of the air inlet because of its curved angle. The air present at the bottom just after the wing can also be seen with a greater velocity.

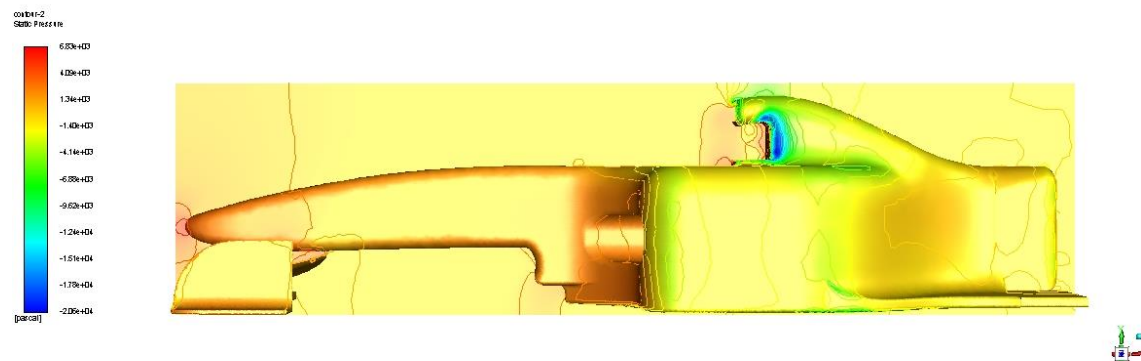
Velocity = 150 m/s



In the above velocity contour we can see that velocity is at a higher speed, hitting the front of the car at 150m/s and then increasing as going forward. From 150 it reached to around 200m/s near the cockpit, and for the air intake it reaches to about 350m/s due to the curvature of the top of the car, this again is an intentional design which helps to transfer more air to the engine at higher speeds.

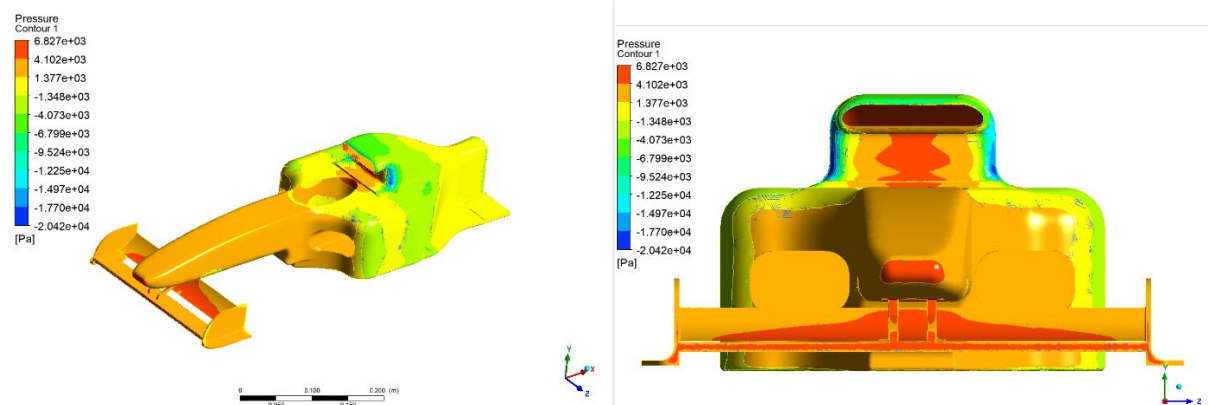
### 3.4 Pressure Contours

Velocity = 50 m/s

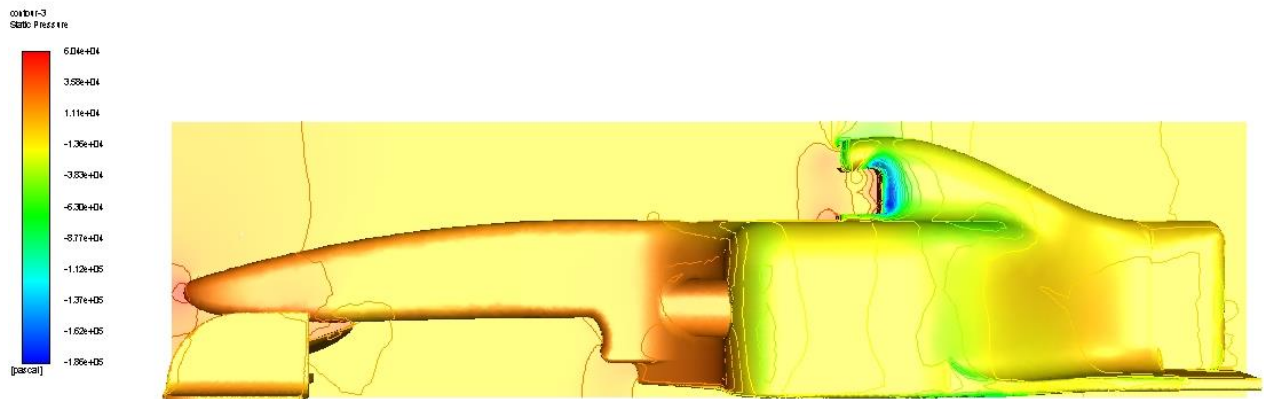


In the pressure contour above we can see that pressure is high at the starting of the nose and near the front wing, whereas it actually decreases a bit near the side air intakes. This mainly helps the car to breathe in more air, as when the pressure decreases the velocity increases, and increasing of velocity near the air intakes means that more air is flowing through the vent which is desired. Pressure decreases a lot to a value of about -0.0015Pa near the sides of the top air inlet.

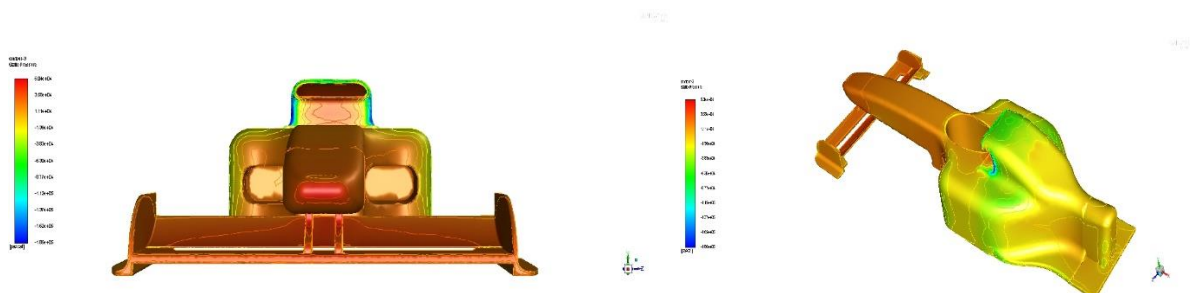
In the contours below we can see that when we take velocity of 50 m/s the pressure at the front wing is actually very high (about 6500Pa). This is highly desirable as it helps in producing more downforce which increases the grip for the tyres and hence improves the performance.



Velocity = 150 m/s



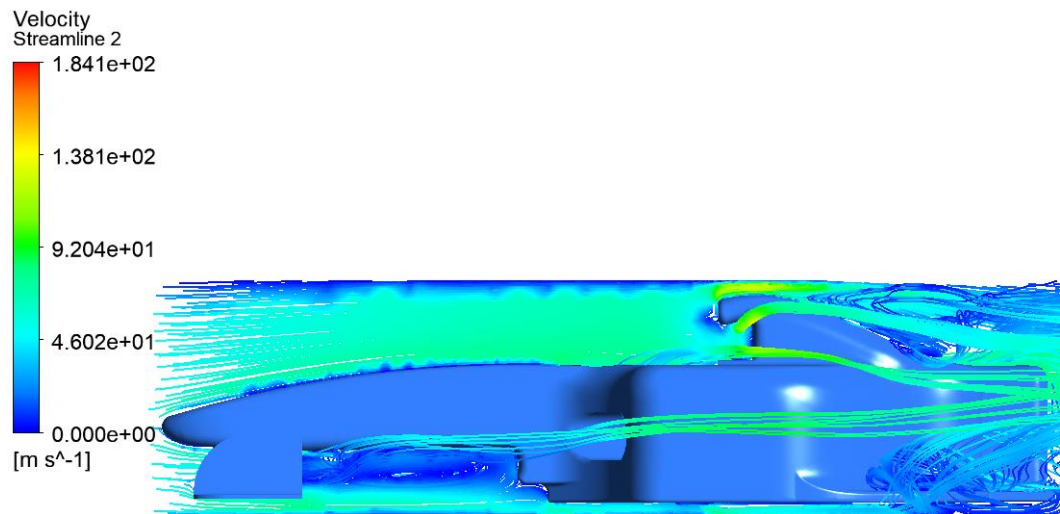
For the pressure contour it can be observed that pressure highest at the front of the car where the wing is placed, it reaches to value of about  $60000$  Pa at  $150$  m/s this generates great amount of downforce which is required at such high speeds. At the air inlets again the pressure decreases to about  $-60000$  Pa which causes the air to have increased speed which is again a desired property.



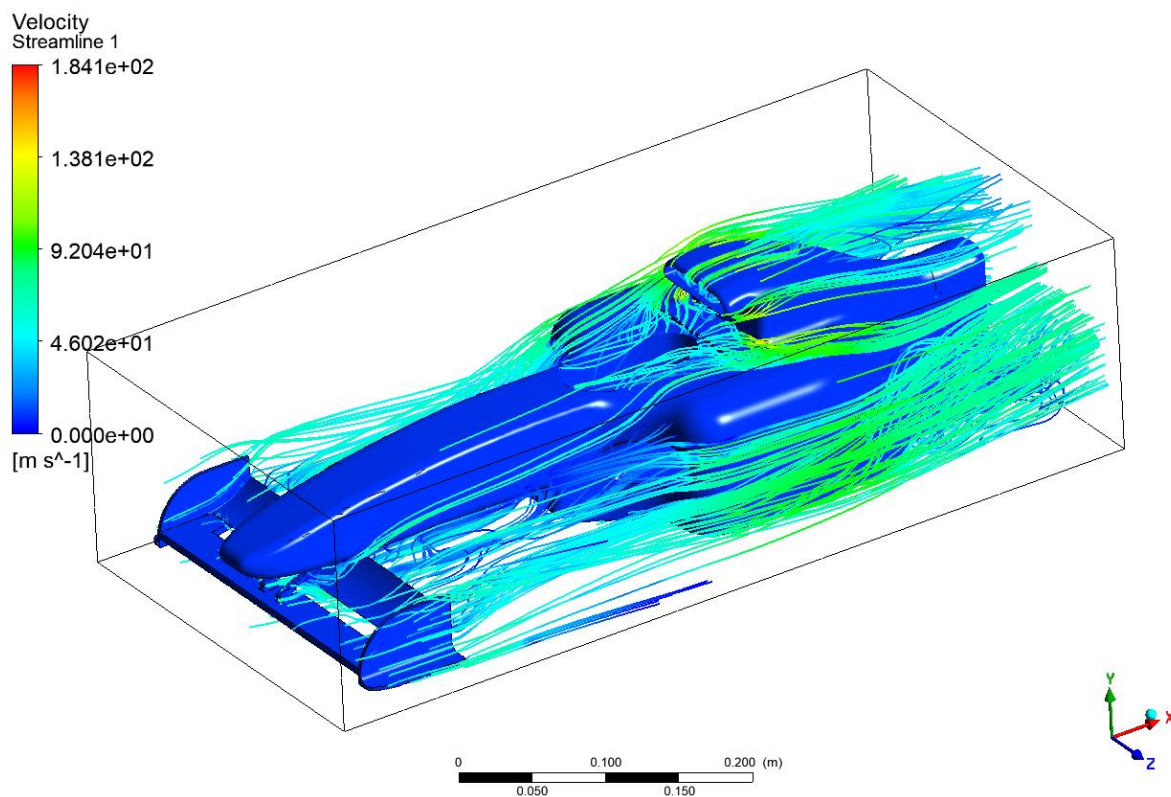


### 3.5 Velocity Streamline

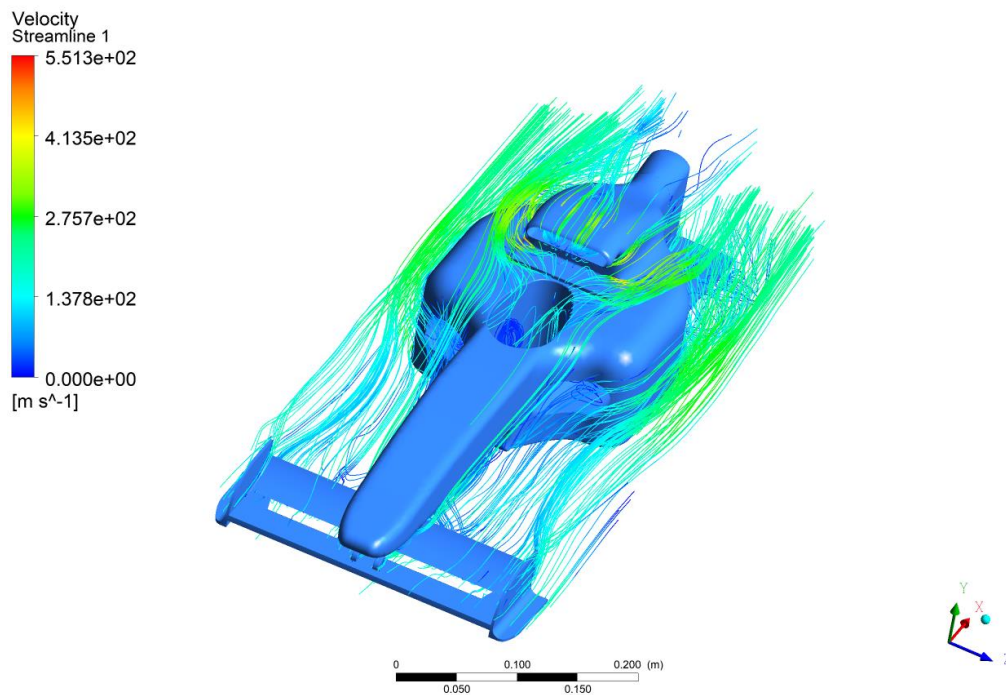
Velocity = 50 m/s



In the velocity streamline we can see that the air cut through the front wing without creating any obstruction but increasing the pressure. While at the air inlets velocity increases, which again helps to supply more air to the engine. At the back of the car we can see that the air is circulating and creating vortices with very low velocity.

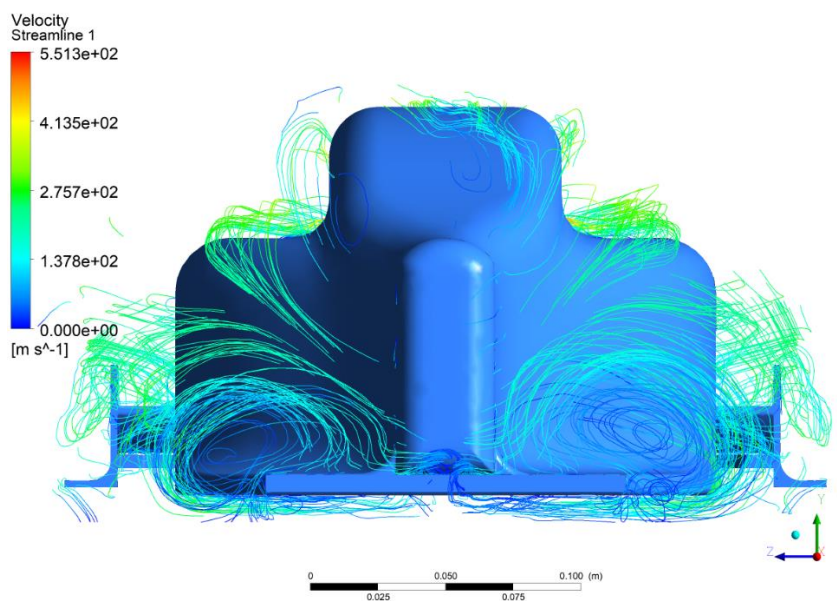


Velocity = 150 m/s



In the velocity streamline for 150m/s it is observed that there is not much disturbance at the front wing, as it is designed to cut through, and the lines that pass from the side of the wing are directed towards the air inlets, where the velocity increases to about 275m/s. At the top inlet air reaches around 450m/s around the corners.

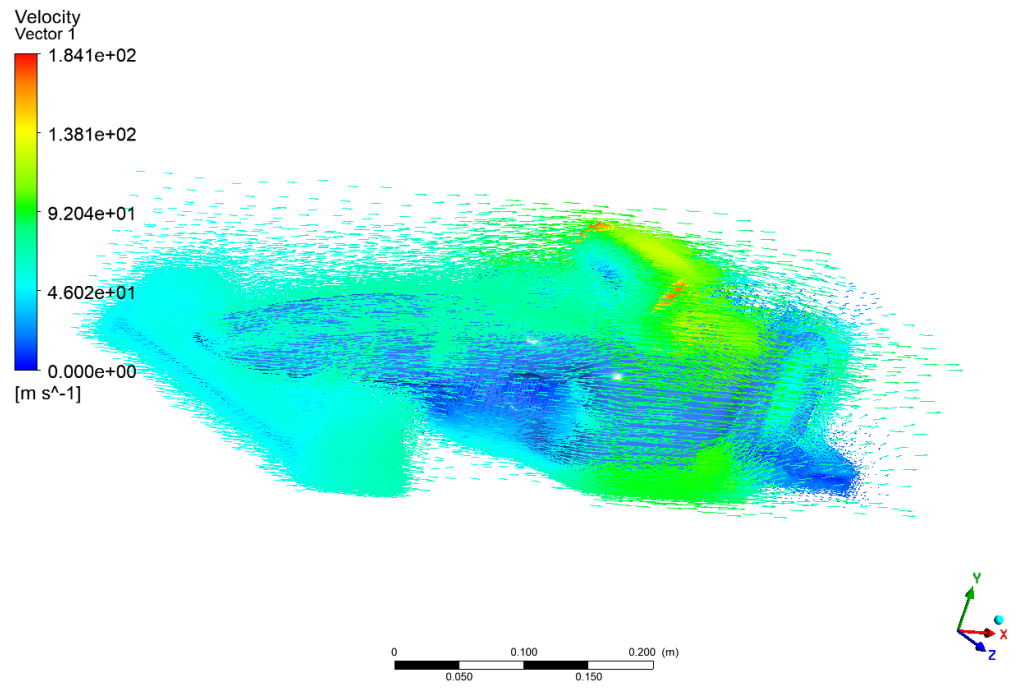
In the figure below we can see that lines form a circular pattern or vortices with decreased velocity, this will again lead to increased pressure at the rear tyres.





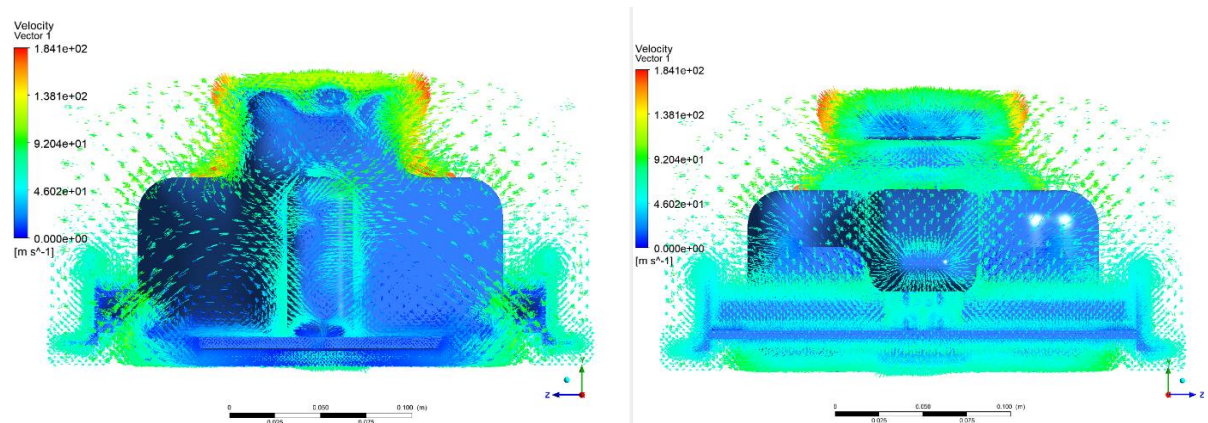
### 3.6 Velocity Vectors

Velocity = 50 m/s

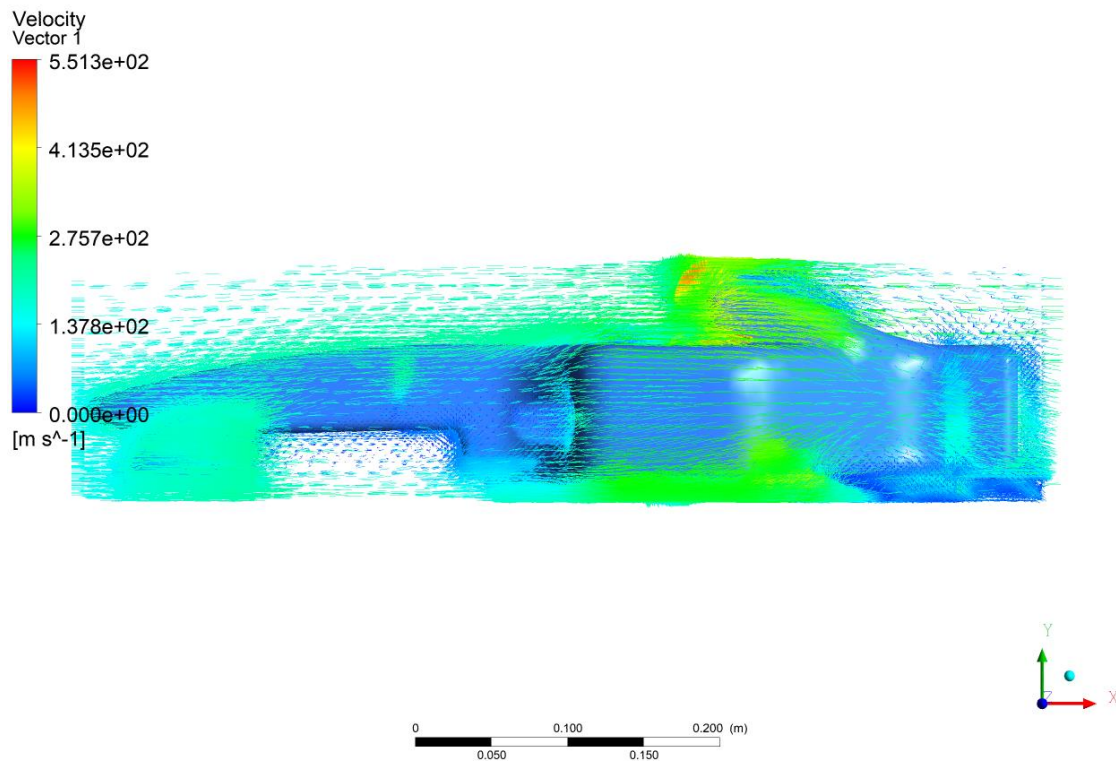


In these figures we can see the velocity vectors represented on the form of 3D arrows with their direction towards the flow of air. The vectors are more concentrated at the front wing with low velocity. They are also concentrated at the air intakes with high velocity.

In the figure below we can see the back of the car and it was noticed that vectors circulates while coming from the sides and then form vortices with very low velocity.

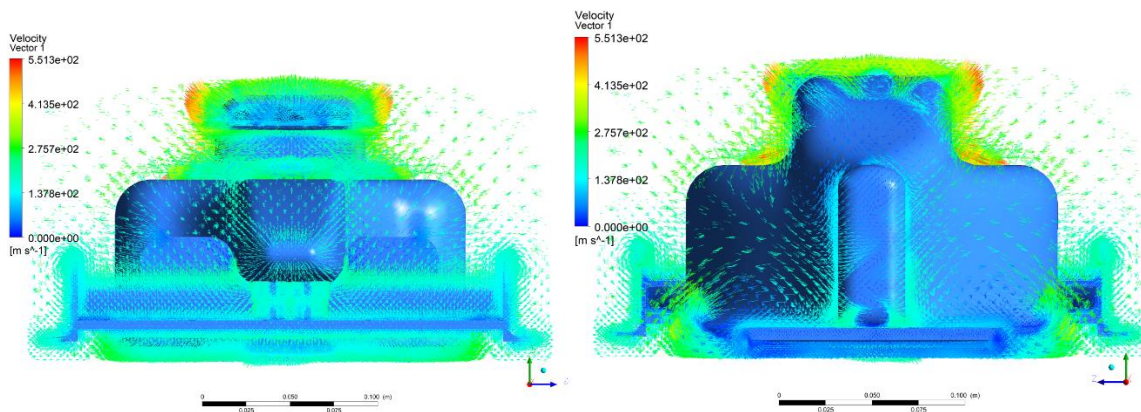


Velocity = 150 m/s



In the velocity vectors we can observe that the arrowheads have a higher concentration at the front wing, bottom of the car and on the top. When velocity is increased to 150m/s the top area of the car is affected more and show change from 50m/s. There the vectors reach to value of about 550m/s.

Where as at the back again vectors move in a circular manner with very low value.



## 4. Discussion and Conclusion

The overall solution represents basic structural working of a Formula1 car, along with this it also proves that different sections of the body and their design is successful for gaining desired aerodynamic properties. Each curve and every bend in all the parts of the F1 chassis is made to contribute to the performance of the car under low and high velocity. The wings are made in such a way that when air hits them, pressure is increased, and this results in increasing the down force. Further after passing over the wing, air goes through the air intakes. The curves on the outside of the vents are designed to increase the velocity which will result in more air flowing to the engine.

When air was at 50m/s the pressure was about 6500Pa at the front and it increased to 60000Pa when air speed was increased to 150m/s. This again is the proof of the concept that it will still perform better under high velocity as well. The same type of result can be observed with velocity contour as well, where maximum velocity reached was about 550m/s. This difference between the outcome values depending upon the initial air flow value shows the conceptual proof of the aerodynamics engineering concepts. The car is performing equally better at higher speeds as it is at lower speeds.

The overall experience for this individual project was really good, I am really impressed by things I have learned while solving this problem, as there was a lot of material that had to be read in order to start from scratch and end up with the complete simulation. Starting with the CAD modelling of the F1 car, various new concepts were unfolded along with the problem that engineers actually face in the real industrial situation. Further analysing the model and the results gave a great deal of insight on how the aerodynamics of the car perform in real time and why are they so beneficial.