Modeling and Computational Fluid Dynamic Analysis of F1 Front-Wing using ANSYS

M. Pavan Kumar[1] Md. Muzammil[2] M. Naga Sai[3] A.Suresh[4]
[1,2]UG Student [3,4]Assistant Professor
[1,2,3,4]Department of Mechanical Engineering
[1,2,3,4]CVR College of Engineering, Hyderabad, India

Abstract— This paper is an outcome of the activity of work for modeling and CFD analysis of front wing F1 car. The configuration of the F1 front wing is done using ANSYS software. Using the ANSYS software the modeling and analysis of computational fluid dynamics is performed. As it's a known thing that F1 cars have the high speed to move on the roads. Hence the drag and lift forces are emphasized for understanding. During the modeling process the starting object is developed as a consequence of the basic front wing & then holds only the end & base plates. The 2nd one is high downward force configuration and it consists of other additional elements of the wing. ANSYS is the software that is used for long time and used vividly as user friendly for the engineers and to professionals to accomplish the role of modeling, CFD analysis. The present analysis is used for the generative design and to use the ANSYS for the design and modeling of front wing. In the coming sections below the details will be discussed briefly by the modeling processes and the sources of dimensions & the results of the aerodynamic study. It can also enlist the concepts and outcomes that are required for understanding the multiple facets of aerodynamic effects along the contact surfaces of the F1. Further it also to be believed to have the information belongs to the modeling as well as analysis which is very much useful to the future mob, who have the aspiration to make the best out of the derived outcomes.

Keywords: CFD Analysis, Front Wing F1 Car, ANSYS

I. Introduction

On the highest speed of the F1 car it attains sufficient enough for take-off, if not to the downward force which makes it down. The F1 car usually generates the necessary down ward force to enhance the needed and required drive towards upside and also to down side over the body of the tunnel. Apart from these the majority of the down ward force develops origin and the prime body by around 15% to 20% could be generated within the front wing of F1 [1,2].

The front-wing of the F1 has also processed to be the engineer's work station. It is because of the fact; as compared and conquest with the design of the F1 car the developers were less confined in the design and development of F1 front wing. It means the complete team in association with the superior edge for the engineers [3,4].

The other possible reason in the context of the front wing design modeling and also the analysis. It also encompasses the emphasis required for the every track that needs a smoothening and slightly variable or configured front wing. Tracks along with long and straights have fewer corners which require the less drag force and to achieve the higher and elevated top speeds & the tracks with more and most corners would need the further more down ward force for attainment of the extra grip for the bends in & around. Hence the front wing itself is continuous and constant development along the time horizon. As an example to consider the tracks for the F1 like Monza (GP) of Italy that has the very long straights which constitutes the teams used for the flat wings to enhance the gain of highest and permissible velocities. For the tracks be Monaco that have the narrowed corners and the teams utilize the steeper end wing elements.

II. Modeling

The virgin step is to accomplish and archive the modeling and designing tool for the object to create the F1 front wing. However based on the given complexity of wing's curves, it is chosen to model from the scratch in terms of generative type of shape design. There is also a learning type of curve though which it has highlighted the important lessons of the modeling in another different section [5].

Modeling of F1 front wing from the scrap to the readily usable form has encountered so many no of hurdles and challenges. Being the first modeled and made car F1 for the sporting propose by abiding the regulations. It conveys in such a way that till the teams follow within the given rules and restrictions in the design, it is free to perform whatever it requires. It also conveys that so much of confidentiality about the modeling and designs. However, this kind of work becomes the car which comes along with some of the ideas with the engineers, at the time of plotting the designs of the wings of F1 front wing. Thus experienced engineers have encountered the same type of problems. And after that it had a major number of parameters and associated dimensions derived out from the most possible and achievable manner. These are the limitations because of them only the regulations confined to make the technical and also professional model. For the other dimensions, it is found that the F1 front wing regulations contain more no of guidelines for the design [6].

The front side wing of F1 is modeled along with the required technical and standard ASTM guidelines & it will be analyzed based on the standards. Some other important parameters and dimensions were also shown in the below table and it has provided the links for the main contents and documents.

The F1 front wing has the airfoil structure and upon which the modeling has started. The F1 front wing profile is in the form of aerofoil. While designing it is very crucial to develop and control the down ward force that is to be developed over the wing. As it is notified earlier, the teams that have changed depend over the type of track and also it is modeled as per the mine to observe the other cross direction of front sie wing of airplanes [7,8].

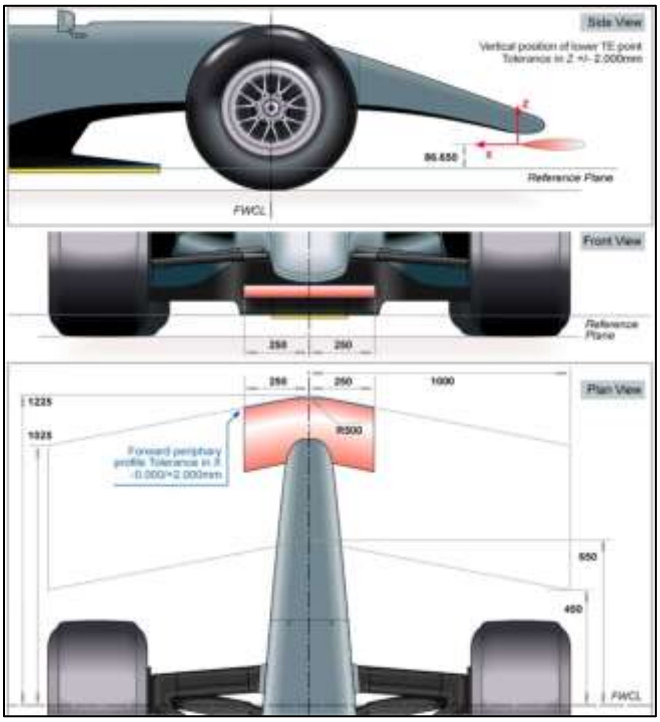

Fig. 1: Dimensions and regulations document

The elements that were functioned are elucidated in detail in the below sections.

The below are the some parameters for developing the actual final product that is to be emphasized. Among the all 1st two used to be simple & robust models then next two were the combination of complex and typical model in association to many other extra elements which are needed for F1 front wing. It is that version, in which the team could make use of what is required. As contrasted to the figures above that visualizes the once in market currently. [9,10]

A. *Assumptions and Preparatory Conditions*

The following assumptions are made for the analysis of front wing using CFD

- The steady state space is to be assumed
- Operating pressure of the ambient conditions is 101kPa
- Gravitational acieration is assumed to be constant
- The time is assumed to follow the k-e model
- The air is assumed to be ideal
- The wing is a symmetric element hence modeling one side will reflect the other part of the wing, however the complete wing is modeled but consideration is made as per the requirement.
- The air is assumed to be incompressible until the speed of the F1 is less than Mac velocity.
- The flow is a combination of laminar and turbulent depends on speed variation.
- The working fluid is air and it has the normal properties as per the air.
- Mesh independence is the one depends on part to part. As the nodes are generated based on the element size. Hence meshing is an independent activity.
- The decision or understanding of convergence is made based on quantity of the pressure and velocity gradient.
- The flow visualization should be three dimensional and both symmetric and asymmetric.

III. PROCEDURE OF COMPUTATIONAL FLUID DYNAMICS (CFD)

The CFD analysis has turned out to become much trickier and meticulous than that is anticipated. This is because of the cumbersome task of meshing. As it has the access to only limited features of the software it is chosen only up to 5,14,000 cells. Since it is that geometry which has involved in present project with much more complex not an easy going activity above the limit. After overcoming all these bottlenecks with the help of error and trail, it becomes difficult to find out the proper and correct size of the meshing.

The main objective of computational fluid analysis is to perform how best the teams utilize the different geometries and configurations to evolve less or more down force. The following are the results which confines the hypotheses that is by default a bit complex and cumbersome one, which would create more amount of down ward force. The below table enlists the analysis of CFD & the procedure in which it can be needed by the wards to use similar kind of simulations.

Process of CFD	Specifications	Parameters	Values
Topology	3D Rectangular Domain	Height, Width, Length	2,3, 20 m
Mesh	Preferences of Physics	CFD	
	Inflation	Rate of growth	2
		Ratio of transition	0.272
	Sizing	Size of Element	0.92957m
Setup	Precision		Double
	General	Solver	Based on the Pressure
		Time	Steady
	Model	Viscous	k-epsilon (2-eqn)
		Energy	Off
	Materials (air -fluid)	Density	1.225 kg/m^3
		Viscosity	1.7894e-05 kg/m-s
	Boundary limitations	Inlet Velocity	117 m/hr
		Pressure at Outlet	0
	Methods	Momentum	2nd order upwind

Solution	Monitors	Drag Force (Z Direction) Convergence tolerance,	1.00E-05
	Initialization	Hybrid	
	Run calculations	Number of iterations	300
Results	Vectors	Velocity	
	Contours	Velocity	Stream function
			Vorticity
			Velocity Magnitude
	Pathlines	X	Velocity, Particle ID, Turbulence
		Y	
		Z	

Table 1: CFD analysis details

A. *Analysis of Results*

Here are some of the outcomes of CFD. The basic and simpler one derives the down force of around 1400N, while many more other complex one develops a down force of around 3100N.

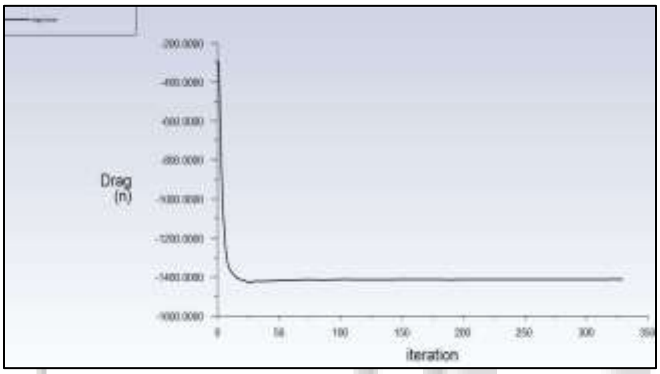

Fig. 2: 1400N Drag type force acting downwards in the Z-direction (simple version)

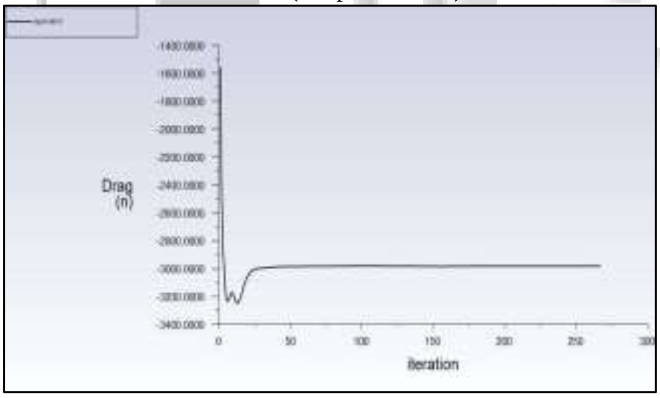

Fig. 3: 3100N Drag-force acting towards Z-direction downwards (complex version)

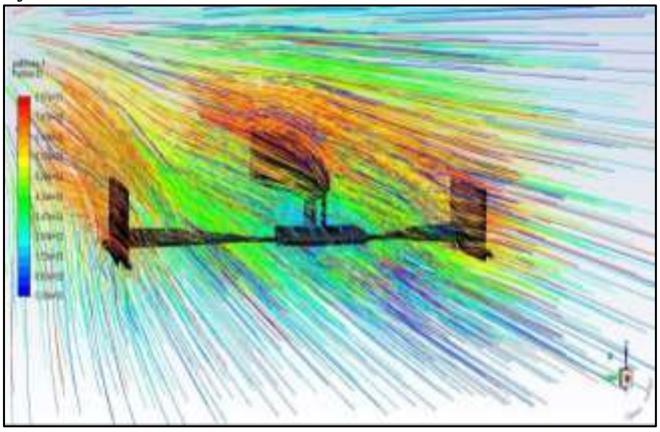

Fig. 4: Path-lines of the front wing CFD in simple version without mixing of the flows

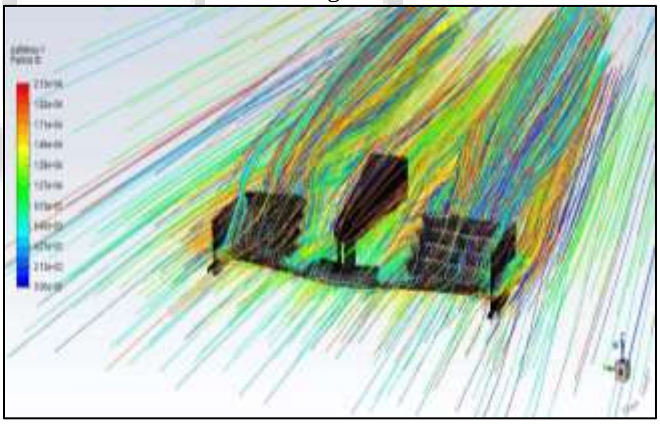

Fig. 5: Path-lines of the front wing CFD in complex version with mixing of the flows

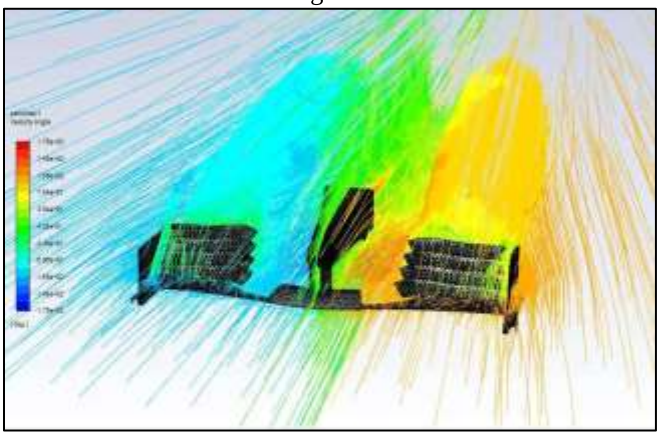

Fig. 6: Kinetic-energy and turbulence in wake

IV. RESULTS AND IMAGES OF SIMULATION

A. *Path-lines*

Path-lines were the lines that travel with the neutrally based buoyant particles within the equilibrium along with motion of the fluid. Path-lines were the best used tools to visualize the complex and meticulous 3D flows.

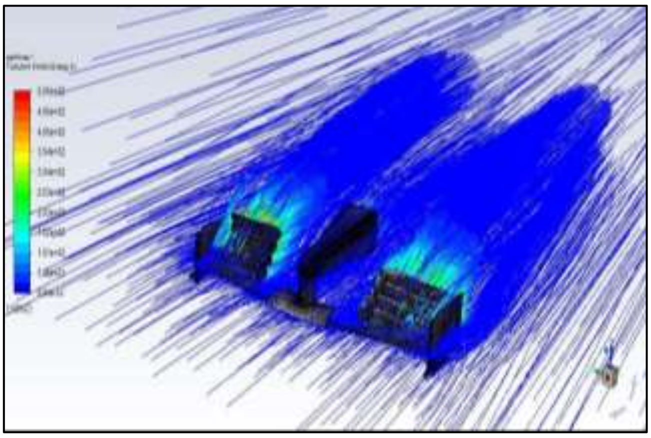

Fig. 7: Profiles of velocity-angles of front wing

V. CONCLUSIONS

A. Overall pressure

The overall pressure over the front wing of F1 is critically analyzed and been represented in the below figure 13. The maximum pressure is been experienced at a velocity of 1.678e002 m/s and the minimum pressure is experienced at a velocity of 0m/s being shown in the figure with saffron and blue colors respectively.

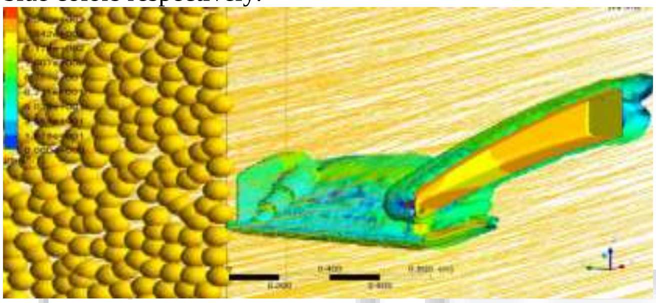

Fig. 5-1: Pressure and Velocity vertices during the speed of 117m/h

B. Drag

The drag force for the F1 must as minimal as possible say it should be optimal. Here in this analysis the drag coefficient is found to be 0.73. i.e., C_d = 0.73.

$C_d = \frac{F_d}{\frac{1}{2}\rho U^2 A}$

$\rho = 1.2255\ kg/m^3$

$A = 1.295\ m^2$

U = 117 mph = 52.303 m/s

F_d = 2337 N

$C_d = \frac{F_d}{\frac{1}{2}\rho U^2 A} = \frac{2337}{\frac{1}{2}*1.2255*52.303^2*1.295} = 0.73$

The drag force acting on the front wing is = 256 N

5.3 Down force

The down force in the analysis has computed as follows.

$F_d = (\frac{1}{2})$*(wing width in m)*(wing height in m)*(angle of wing)*(lift coefficient)*(air density)*(velocity)

$= (\frac{1}{2})*(2)*(3)*(10)*(0.71)*(1)*(52)$

= 1107.6 N

C. Evaluation of the Relative Importance of Various General Flow Mechanisms for Aircraft and Vehicle Aerodynamics

The flow mechanism for the aircraft and vehicle aerodynamics is much relative and it confines to the same community. As the aerodynamics is the analysis of object movement through the air and it makes the flow of the object against the friction. The flow mechanism is the category of types of flow. It is much related to the laminar flow or say turbulent flow or a transition or mixed flow. And in spite of that there is a huge difference in the context of aircraft and other vehicles. All these are detailed in the table no 2.

S No	Parameter	Air craft	Vehicle aerodynamics
1	Speed	High speeds like MAC or above or below	It ranges from smaller value to high values
2	Flow	Compressible	Incompressible and compressible
3	Force	Drag and lift forces along with down ward forces	All the types of forces involved
4	Wings	It must contain wings	Need not to use always
5	Eddy formations	Must form	May or maynot

Table 2: Difference between air craft and vehicle aerodynamics

D. Evaluation of the Appropriateness of Different Analytical, Experimental and Numerical Methods

The evaluation of appropriateness of different methods is given below in table no 3.

S No	Type	Appropriateness
1	Force determination methods along with 2 degrees of freedom model	For limited no of outcomes
2	Multi degree of freedom model	For more no of outcomes
3	Experimentation with model analysis	It is to be used in case of small or medium scale experimentation
4	Experimentation with prototype analysis	It is to be used in case of large scale experimentation
5	Analysis with modeling and CFD software	More appropriate for accuracy in results

Table 3: Methods and appropriateness

E. Application of CFD to Real Flows

The CFD is very much useful and it is an appropriate tool for the flows type of problems. Whenever the object dynamic analysis to be carried out there involves the computation of the flow properties and analysis based on the movement and object orientation. It also applied for the understanding of the vehicle movement by evaluation the pressure heads, velocity heads and also the formation of eddies. It also applied for the objects that are been used extensively in the aerospace field.

REFERENCES

[1] Mohammad Arief Dharmawan, Ubaidillah,brian Aqif Nuafal,et.al, Feb-2018 “Aerodynamic analysis of formula student car”, AIP conference proceedings,1.5024107.
[2] Shreyas Vaidya,Chinmay Kulkarni, ,Decembar2017, “Aerodynamic development of formula SAE car”,International Journal of Engineering Research & Tevhnology,2278-0181.
[3] Francesco Mariani, Lorenzo scappticci et.al., 2015, “Experimental and Numerical analysis of external aerodynamics”, ELSEVIER, ,1013-1029.
[4] Rizal E.M. Naseer, Firdaus Mohamad et. al, 2012, “Aerodynamics of ARTeC’s PEC Emo-C car”, ELSEVIER, ,1775-1780.
[5] Anirudhha Patil ,Siddharth Kshirsagar, 2014, “Study of front wing of F1 car using CFD”, International Journal of Mevhanical Engineering And Robotics Reasearch, ,2278-0149.
[6] Triya nanalal vadgama, Aprit Patel et.al, May-2015, “Structural analysis of F1 car”, International Journal and Research in Engineering,Science & Technology, 2394-2444.
[7] S. Pehan and B. Kegl, 2002, “Aerodynamic Aspect of Formula S Racing Car”, International Design Conference-Design, May-2017.
[8] Aravind Prasanth, Sadjyot Biswal et.al, July-2016, “Complete Design and Optimization of the Aerodynamics of FSAE Car using Solid works, ANSYS, XFLR5”, World Congress on Engineering, , 2078-0958.
[9] Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan, 2011,'Heat Transfer Simulation by CFD from Fins of an Air Cooled Motorcycle Engine under Varying Climatic Conditions', Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, London, U.K.
[10] Mukesh Didwania, Gopal Krishan, Ravikant, April 2013,'Study and Analysis of Heat Transfer through Two Different Shape Fins using CFD Tool', International Journal of IT, Engineering and Applied Sciences Research (IJIEASR) ISSN: 2319-4413 Volume 2, No. 4.