International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 03 | Mar 2021 www.irjet.net p-ISSN: 2395-0072

Performance Analysis of a Formula 1 car using CFD

Nagalla Satya Abhisht[1]

[1]III B. TECH AERO, GITAM UNIVERSITY, Rudraram, Hyderabad, Telangana, India

---***---

Abstract - *This paper presents the modeling and aerodynamic study of two Formula one front wing configurations using CATIA and the ANSYS workbench. The first configuration is a simple front wing only consisting of the base and endplates. The second is a high downforce configuration consisting of additional wing elements. CATIA has for a long time been the choice of software for professionals when it comes to surface modeling. I have used the Generative shape design workbench to model the front wing. In the sections below I will briefly discuss my modeling process, the source of dimensions and the results aerodynamic study. I will also include the lessons I have learnt, and things I would change if I were to do this again. I believe the information will be useful to future students who want to explore this powerful workbench further.*

***Key Words*: CFD, meshing, CATIA, k-ε' model, pressure contour, velocity contour, aerodynamics**

1. INTRODUCTION

Initially the designers of Formula One car were hooked in to the horsepower for achieving their aim of top speed but recently they're trying to realize their aim through aerodynamic forces. The aerodynamic setup for a car can vary considerably between race tracks, counting on the length of the straights and therefore the refore the sorts of corners; and the optimum setup is usually a compromise between the 2. Larsson T et al, studied highly sophisticated body shape of a contemporary Formula One (F1) car, which is dictated by aerodynamic efficiency and performance. With numerous deflectors and external devices integrated to the bodywork, understanding of the coupling and interaction between the front-end and rear-end of the car has become vital. Minute changes in geometrical details of those components can often have a worldwide impact on the general flow topology, therefore influencing car performance. Tamás Régert and Dr. Tamás Lajos, studied the effect of rotating wheels on the flow past cars. The flow fields past wheels influence the cooling of brakes, underbody flow, mud deposition, drag and lift forces working on the car body. Kyoungwoo Park et al, performed numerical analysis for the aerodynamic characteristics of the wing-in aerodynamic lift craft with highly cambered and ratio of 1 to predict the bottom effect for the case of with and without lower-extension endplate. The analysis included varying angles of attack from 0 to10 degree and ground clearances from 5% of chord to 50%. Satyan Chandra et al, have performed an analytical study on air flow effects and resulting dynamics on the PACE Formula-One racer. This study incorporates Computational Fluid Dynamic analysis and simulation to maximise down force and minimize drag during high speed of the racer. The study also includes optimization of wing orientations (direct angle of attack) and geometry modifications on outer surfaces of the car are performed to reinforce down force and lessen drag for max stability and control during operation. F Mortel, has studied different aerodynamic devices attached to the front wing of the Formula-One car. This study has shown necessity of endplates to scale back drag due turbulence created at the top of the wing. Joseph Katz, has discussed typical design tools like structure testing, computational fluid dynamics, and track testing, and their relevance to racer development. This review briefly explains the importance of the aerodynamic down-force and the way it improves racer performance.

1.1 Computational fluid dynamics

Computational Fluid Dynamics (CFD) may be a computer-based technology that studies the dynamics of all things that flow. In Formula One racing, CFD involves building a computer-simulated model of a racer then applying the laws of physics to the virtual prototype to predict what the down-force or drag could also be on various components of the car or how the car will respond in various wind conditions, changing environmental conditions or on different road surfaces. Features of the matter to be solved, the essential procedural steps are given below:

- Define the modeling goals.
- Create the model geometry and mesh.
- Find out the solver and physical models. Study of Front-Body of Formula-One Car for Aerodynamics using CFD
- Compute and monitor the answer.
- Examine and save the results.
- Consider revisions to the numerical or physical model parameters, if necessary. Because the Formula One car average speed is around 55-60 m/s (around 200 km/h), the Reynolds number handling the front wing are from 106 to 3x106 (depending on the device picked on the front wing). Therefore, the flow would definitely be a turbulent one at the front wing surfaces. Thus a 'k-ε' model has been selected for analysis.

1.2 k-ε' model

The k-ε (k-epsilon) model is the industry standard two-equation turbulence model. k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L2 T-2), e.g., m2/s2. ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of k per unit time (L2 T-3), e.g., m2/s3. The k-e model, like the zero-equation model, is based on the eddy viscosity concept,

It assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation as below

For turbulent kinetic energy k[4]

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon$$

For dissipation ε[4]

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

2. Realizable k-ε' model

The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. To understand the mathematics behind the realizable k - ε model, consider the following expression for the normal Reynolds stress in an incompressible strained mean flow.

$$\varepsilon_{\mathrm{ij}} = \frac{2}{3}\varepsilon\delta_{ij} \text{ or } e_{\mathrm{ij}} = \sigma a_{ij}$$

$$\text{where } a_{ij} = \frac{\overline{u_i u_j}}{k} - \frac{2\delta_{ij}}{3} = 2b_{ij}\,.$$

3. MODELLING OF F1 CAR

The model of the car is designed in CATIA as per the given technical regulations for a F1 car by FIA.

First of all, a model without endplates has developed to see endplates as flow deflectors. 'Basic Models' are created with simple wing hanging onto nose and endplates attached to the wing ends. Then three models are developed from the essential Model with optimum angle of attack by attaching additional down-force productive plates to the endplate

This primary design sets a Basic Model of the front wing. Point of the front wing on the vanguard is 950mm, from the wheel center as per the regulations. Base plane is taken at a height of 100mm from road.

Models are created by changing the angle of attack (α) from -60 to -100 so on select optimum angle of attack for the essential Model. To deflect the be due wheels so on reduce drag, a vertical wing (endplate) is introduced. Thickness of endplate is 32mm. it's assumed that the flow should remain attached on the side of the wing.

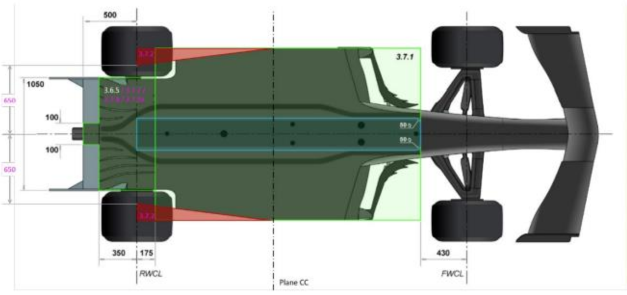


Fig 1: Dimensions of car

3.1 CATIA Model

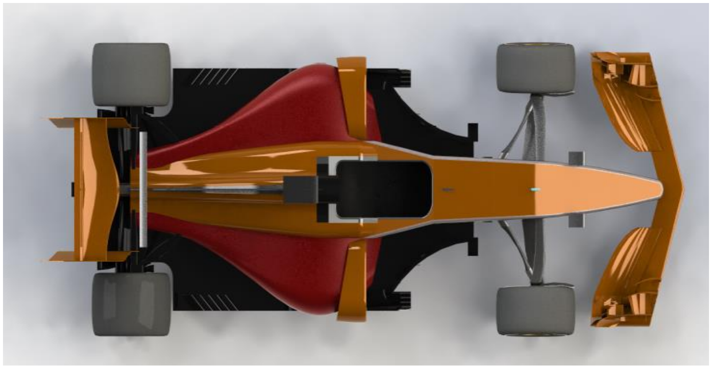


Fig 2: CATIA model

3.2 Meshing requirements

The car body is symmetric around a few central vertical planes, thus only half portion is employed for analysis. As flow round the car model had to be analyzed, an enclosure of certain volume has been created around the model, in "Fluent". "Boolean" operation has been used to subtract car model from enclosure. Remaining portion has been used for flow analysis.

Mesh parameters has been maintained as below:

- Minimum element size = 5mm
- Aspect ratio = 1 to 14
- Average skewness = 0.2

3.3 Boundary conditions

The boundary conditions are set as follow:

1) The inlet of the domain: The air speed at the doorway, "Inlet velocity" = 60 m/s.

2) The front body-work: Set as a "wall" – stationary.

3) The ground: Set as a moving "wall" - Its speed is adequate to the air velocity (60 m/s).

4) The wheel: Set a moving "wall" - The wheel was built as a cylinder and rotates around its axis at 181.82 rad/s in order that it's adequate to a speed of 60 m/s on the track.

5) The outlet of the domain: Set as "outflow" – outlet pressure is about as air pressure. air pressure = 101325 Pa. Gauge pressure = 0 Pa.

6) The edges of the domain: Set as "symmetry" - Side that represents center of the car because the latter was split into two sections.

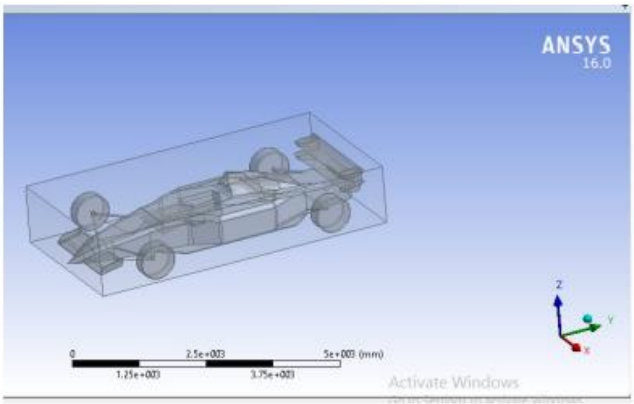


Fig 3: meshing

4. RESULTS AND DISCUSSIONS

The analysis of the model is done using the FLUENT software in ANSYS. The number of iterations has been 1000 for the particular model maximum pressure and maximum velocity at the centerline have been obtained different flow contours and Cl and Cd have been calculated.

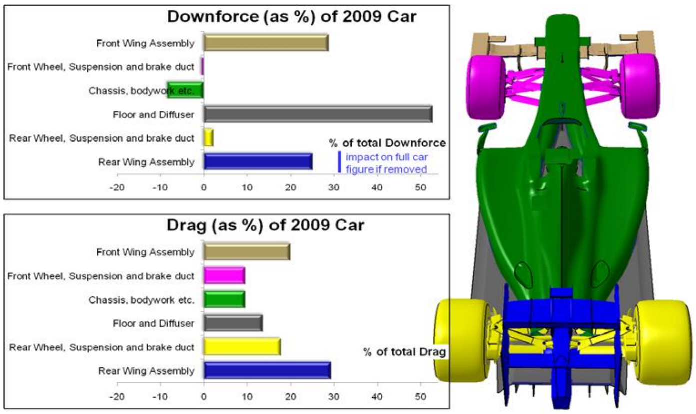


Chart 1: estimated drag and down force

4.1. Contours

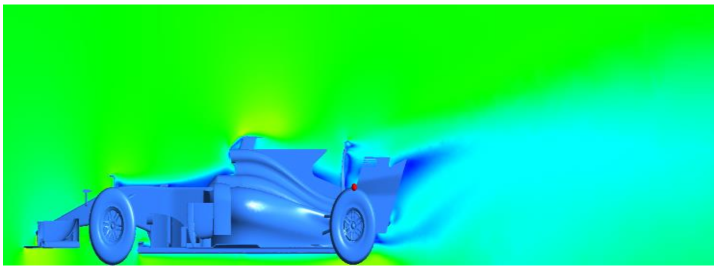


Fig 4: velocity contour

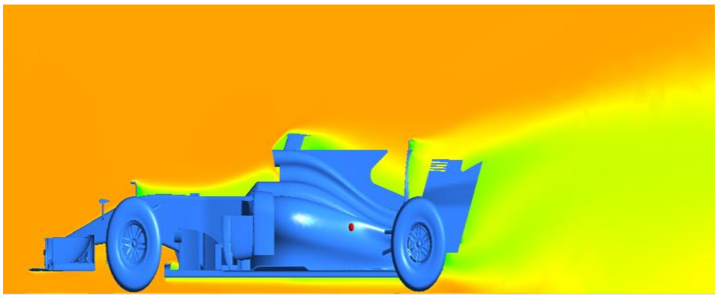


Fig 5: pressure contour

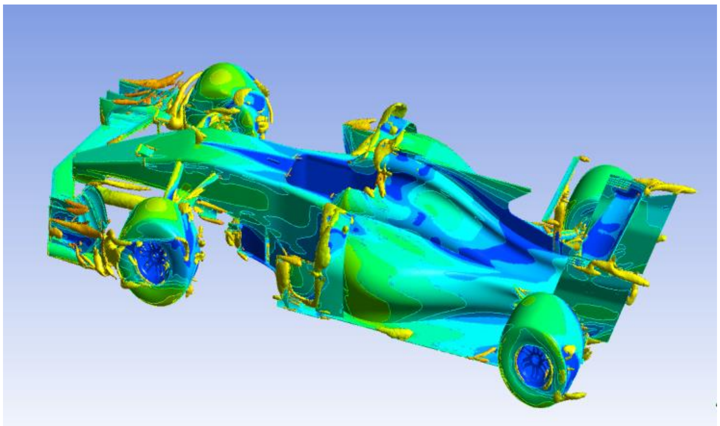
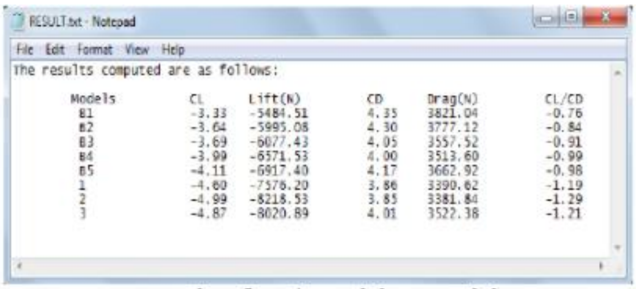


Fig 6: analysis of car

4.2 Performance



RESULT.txt - Notepad

File Edit Format View Help

The results computed are as follows:

Models	CL	Lift(N)	CD	Drag(N)	CL/CD
B1	-3.33	-5484.51	4.35	3821.04	-0.76
B2	-3.64	-5995.08	4.30	3777.12	-0.84
B3	-3.69	-6077.43	4.05	3557.52	-0.91
B4	-3.99	-6571.53	4.00	3513.60	-0.99
B5	-4.11	-6917.40	4.17	3662.92	-0.98
1	-4.60	-7576.20	3.86	3390.62	-1.19
2	-4.99	-8218.53	3.85	3381.84	-1.29
3	-4.87	-8020.89	4.01	3522.38	-1.21

Fig 7: cl and cd

Different models have been created and cl and cd for them have been calculated for basic models.

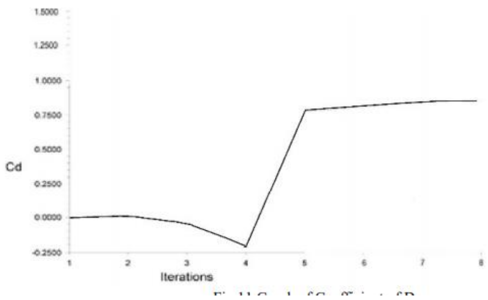


Fig 8: graph of Cd

4.3 Flow over wheels

Due to rotation of wheels vortices are created at the ground surface where the wheel is in contact with the ground. So it forms a low pressure region at the ground surface

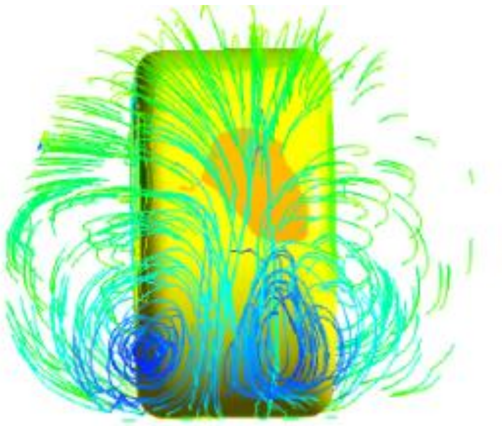


Fig 8: flow near ground surface of wheel

5. CONCLUSIONS

1. Aerodynamics plays an important role in the design of any vehicle.

2. Here, in this paper, an attempt was made to create an F1 racing car model and to analyze the stability of the car including the drag produced during high performance.

3. Using CATIA V5 software, we generated an F1 racing car model.

4. Using Ansys 16.0 workbench we have analyzed the stability of the vehicle.

5. The coefficient of drag of formula 1 car is between 0.7 to 1.1, in our study, at the velocity of 60m/s, the coefficient of drag was found to be 0.85 of the modeled Formula 1 cars.

REFERENCES

[1] https://cfd2012.com/ansys-cfx-formula-one-car.htmlM. Young, The Technical Writer's Handbook. Mill Valley, CA: University Science, 1989.

[2] Manoj Kumar D. Birajdar, Suresh Choudhary and Prof. Vivek Mane. 2017. "CFD Analysis of an Automobile to Improve the Aerodynamics", International Journal Of Advance Scientific Research And Engineering Trends. (September 2017), Volume 2, Issue 2. ISSN: 2456-0774.

[3] https://www.google.com/url?sa=i&url=http%3A%2F%2Fwww.wseas.org%2Fmultimedia%2Fjournals%2Ffluid%2F2019%2Fa125113-243.pdf&psig=AOvVaw2LmlGniPVBqj6tHT_z5UvS&ust=1616653325203000&source=images&cd=vfe&ved=0CA0QjhxqFwoTCJiNv5GlyO8CFQAAAAAdAAAAABAK.

[4] A. Muthuvel,and N. Prakash. 2014. "Numerical Simulation of Drag Reduction in Formula One Cars", J. Godwin John International Journal of Engineering Research and Applications. (29th March 2014), ISSN: 2248-9622.

BIOGRAPHIES

Nagalla Satya Abhisht currently pursuing III B.Tech at GITAM University, Hyderabad. My research areas are computational fluid dynamics, data mining, and propulsion in aerospace