Proceedings of the 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece, August 21-23, 2006 (pp29-34)

FINDING THE OPTIMUM ANGLE OF ATTACK FOR THE FRONT WING OF AN F1 CAR USING CFD

J. Jagadeep Reddy
B. Tech (Mech) IIIrd year
VIT, Vellore-14(TN),India

Mayank Gupta
B. Tech (Mech) IIIrd year
VIT, Vellore-14(TN),India

Abstract:-The F1 car is vehicle designed to obtain maximum speed across a race track. Earlier, the main mode for achieving speed was the development of engine but now aerodynamic forces – downforce and drag – are an object of concern by the team to achieve higher speeds. The 'drag' and 'downforce' are the two important forces governing the efficiency of a road vehicle. They influence the top straight line speed and cornering speed significantly for an F1 car. This in turn influences the performance of the car. The general design of the vehicle is such that lot of downforce is required to keep the car glued to the track.. The front wing, rear wing and the diffuser are the important components to achieve this.
The front wing is supposed to generate about 25% of this 'downforce'. These forces are dependent on C_L & C_D which depend on the angle of attack. The paper uses a numerical approach to finding the variation of these parameters on angle of attack using the CFD software FLUENT. In the meshing software 'GAMBIT' the boundary conditions for the problem were specified as per the real problem analysis. The Reynold's number for this kind of flow is between 10^6 to $3*10^6$. Hence, 'k-ε' model of turbulence was used. The results were correlated with previous results. Subsequently, the angle of attack was altered for obtaining the parameters at various angles to obtain the optimum angle of attack.

Keywords:
CFD – computational fluid dynamics, CAE, Reynold's number, downforce

1. Introduction:-

The main objective of the F1 teams is to achieve top speed. Earlier the designers were dependent on the horsepower for achieving their aim but recently they are trying to achieve their aim through aerodynamic forces. The aerodynamic setup for a car can vary considerably between race tracks, depending on the length of the straights and the types of corners; and the optimum setup is always a compromise between the two.
The wings fitted to these cars are very significant factors of aerodynamic forces. Negative lift is induced by creating a lower pressure below the wing which is created by higher-velocity airflow below the wing surface. This negative lift comes at a cost. For any amount of lift gained, drag also increases. The drag forces are an important factor in determining the attainable top speed. These forces are determined by the angle of attack set up by the front wing as it is the first part of the car to come in contact with the air.

2.1 Literature:-

Angle of attack is a term used in aerodynamics to describe the angle between the airfoil's chord line and the direction of airflow wind, The amount of lift generated by a wing is directly related to the angle of attack, with greater angles generating more lift and more drag. This remains true up to the stall point, where lift starts to decrease again because of airflow separation. A minor change in angle of attack or height of the vehicle has caused the car to

experience lift, not downforce, sometimes with disastrous consequences.[1]
The goal of any designer in the wind tunnel is to maximize negative lift while also minimizing drag. The greater the Lift-to-Drag ratio, the faster the lap times ratio, the faster the lap times are. [2]The wind tunnel provides an effective means of simulating real flows. Recent works have shown the variation of C_l & C_d with the angle of attack of high performance vehicle [3] for wind tunnel testing. In the design of equipment that depends critically on the flow behavior, like the aerodynamic design of an aircraft, full-scale measurement, as part of the design process is economically impractical. This situation has led to an increasing interest in the development of a numerical wind tunnel. Costs incurred by the F1 companies in running the wind-tunnels have been tabulated for the top four F1 teams in the year 2005

Table 1.
Wind Tunnel Costs[4]

	Company name	Cost incurred in wind tunnel testing in 2005(in millions dollars)
1.	Ferrari	11.25
2.	Mclaren-Mercedes	9.96
3.	BMW-Wiliiams	9.76
4.	Toyota	8.95

This situation has led to an increasing interest in the development of a numerical wind tunnel. In [5], the author has calculated the C_l and C_d for an airfoil by simulating the boundary layer suction theory with a aerodynamic design program in the FORTRAN source code.

2.2 CFD Review:-

The Physical aspects of any fluid flow are governed by three fundamental principles: Mass is conserved; Newton's second law and Energy is conserved. These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. This branch of fluid dynamics complements experimental and theoretical fluid dynamics by providing an alternative cost effective means of simulating real flows.

Computational Fluid Dynamics is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. This branch of fluid dynamics complements experimental and theoretical fluid dynamics by providing an alternative cost effective means of simulating real flows. As such it offers the means of testing theoretical advances for conditions unavailable on an experimental basis.
CFD technology is now mature enough to provide sufficiently accurate results for the external aerodynamic analysis. Choice of cell shapes, mesh structures and grid resolution, influences the quality of the CFD results more than any other single factor. Most automatic mesh generation strategies use tetrahedral cells that are highly diffusive, requiring very large number of cells to produce accurate results. It is not unusual for an all-tetrahedral mesh cells to accurately predict flow around an F1 racing car. Other issues affecting the quality of results include the choice of turbulence models, choice of wall treatment, boundary positions and conditions, and assuming whether flow around the car is steady or transient. [8] F.Mortel in his thesis [6] in the year 2003 has shown the emphasis of different designs of airfoils for achieving aerodynamic efficiency. The C_l variation with the angle of attack for a finite wing has also been shown. [7]

2.2(a) Airflow Modeling [9]

Airflow modeling based on Computational Fluid Dynamics (CFD), for the fundamental conservation equations for mass, momentum and energy in the form of the Navier-Stokes equation, are:-

$$\partial/\partial t(\rho\varphi) + div(\rho\vec{v}\varphi - \Gamma_{\varphi} grad\varphi) = S_{\varphi}$$

Transient + Convection- Diffusion=Source
Where, ρ = Density
$\vec{v}$ = Velocity Vector
φ = Dependent variable
Γ_{φ} = Exchange Co-efficient(Laminar+ Turbulent)
S_{φ} =Source and Sink

Airflow modeling solves the set of Navier-Stokes equations by superimposing a grid of many tens or even hundreds of thousands of cells that describe the physical geometry and heat sources and air itself.

2.2(b) 'k-ε' model

The **'*k-ε*'** model is derived by substituting the sum of an average term plus a fluctuating term for he instantaneous quantities in the equations below:-

$$\frac{\partial p}{\partial t} + \nabla \rho u = 0$$

$$\rho(\frac{\partial u}{\partial t} + \frac{1}{\alpha}\nabla|u|^2 - u \times w) + \nabla p - \rho g = \nabla \sigma$$

The average terms are expected to vary less than the instantaneous quantities and, therefore, can be resolved over a coarser grid. This averaging procedure yields an additional unknown term called the Reynold's Stress ($-\rho u_i' u_i'$). The additional unknowns are resolved by introducing the eddy viscosity concept, which results in two additional transport equations, one each for **'*k*'** and **'ε',** and five empirical constants.[10]

For the two-equation model based on both a transport equation for turbulent kinetic energy *k* and a transport equation for the dissipation of turbulent kinetic energy ε.

A general formulation is given by [11]

$$\frac{D\rho\varepsilon}{Dt} = \frac{\partial}{\partial x_i}((\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}) + P + G - \rho\varepsilon - C_5 2\mu(\frac{\partial\sqrt{K}}{\partial x_i})^2$$

Equation 1

$$\frac{D\rho\varepsilon}{Dt} = \frac{\partial}{\partial x_i}((\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_i}) + C_1 f_1 \frac{\varepsilon}{k}(P + C_3 G) \quad \text{Equation 2}$$

$$-C_2 f_2 \rho\frac{\varepsilon^2}{k} + C_4 \frac{2\mu\mu_t}{\rho}(\frac{\partial^2 u_i}{\partial x_i \partial x_j})$$

Where $P = \mu_t \frac{\partial u_i}{\partial x_j}(\frac{\partial u_i}{\partial u_j} + \frac{\partial u_j}{\partial x_i})$

And the buoyancy term, $G = \beta g_i \frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial x_i}$

The turbulent viscosity μ_t is obtained from

$$\mu_t = \rho c_\mu f_\mu \frac{k^2}{\varepsilon} \qquad \text{Equation 3}$$

The conventional *k-e* model is achieved when f_1, f_2, f_μ and C_3 are equal to one and C_4 and C_5 are zero. A low Reynolds number **'*k-ε*'** describing flow close to a solid surface can be obtained from equations 1,2 and 3 using the following expressions

$$f_1 = C_3 = C_4 = C_5 = 1.0$$
$$f_2 = 1.0 - 0.3 \exp(-R_t * R_t)$$
$$f_2 = \exp(-3.4/(1 + R_t/50)^2)$$

where the turbulent Reynolds number is

$$R_t = \rho k^2/\mu\varepsilon$$

The iteration procedure in a numerical prediction often will be stabilized by a high level of turbulence. Initial iterations can be performed with a high and constant eddy viscosity and the prediction, at a later stage, be connected to a *k-e* model.

3. CAD Model:-

3.1 Front Wing-

Model of the front wing was designed in the 'SOLIDWORKS 2005' as per the regulations for the year 2005' and considering all the above mentioned factors influencing the aerodynamic properties. The relevant regulations are tabulated below in table 2

Table 2[12]
Article FIA Formula 1 Technical regulation

1.	**Overall height**
	No part of the bodywork may be more than 950mm above the reference plane.
2.	**Front bodywork height**
	All bodywork situated forward of a point lying 330mm behind the front wheel centre line, and more than 250mm from the centre line of the car, must be no less than 100mm and no more than 300mm above the reference plane.
3.	**Bodywork around the front wheels**
	With the exception of brake cooling ducts, in plan view, there must be no bodywork in the area formed by two longitudinal lines parallel to and 400mm and 900mm from the car centre line and two transversal lines, one 350mm forward of and one 800mm behind the front wheel centre line.

The wheel was designed with a radius of 330mm, there wheel were cut on both faces with a circle of 100mm. radius for a depth of 100mm. to make space for the suspension rods on the inside and for the wheel mountings on the outside. The suspension rod is used to connect the wheel to the nose of the car.

Also the edges of the wheel were rounded off with a fillet of radius 50mm. to lessen the drag.
The furthermost point of the front wing is 900mm. from the wheel center as per the regulations and the end plates are 200mm. thick. The sweep back angle used for the wing is 5 degree. The chord length of the wing is 200mm. and it is extended for 1360mm. between the end plates. The width of the whole front wing is 1400mm. where as the wheel base of the car is 1800mm. The wing main plane is often raised in the center. This again allows a slightly better airflow to the under floor aerodynamics, but it also reduces the wings ride height sensitivity.[13]
Each front aerofoil is made a main plane running almost the whole width of the car suspended from the nose. The flaps are usually made of one piece of carbon fiber. On each end of the main plane there are endplates.The end plates are 200mm. thick The primary function of the end plate is to stop the high-pressure air on the top of the wing from being encouraged to roll over the end of the wing to the low-pressure air beneath, causing induced drag. Also, the design aim of the endplates is to discourage the dirty air created by the front tire from getting under the floor of the car.

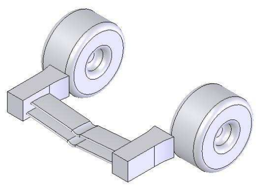

Figure 1

If angled, the vane can generate more downforce as air flows over the top surface more quickly than it does over the lower surface. This gives the car greater stability during cornering while reducing straight line speed.[14]. The main aim is to deflect the sir over the car as under the car the air faces a lot of obstruction to it's flow and can not be smoothened to a greater accuracy compared to over it.

3.2 Meshing-

The created model in 'SOLIDWORKS' was exported to the 'GAMBIT' for meshing. In 'GAMBIT', as the model is symmetric about it's centre line, hence, it was split in halves about the centre line. To apply the Finite Volume Approach a volume [1200x1000x2000] was created around the split model where the boundary conditions were defined. The wheel base was made to coincide with the floor of the volume. The volume created was meshed separately than the wheel and wings.
The meshing used was Quadrilateral\Pave for with a spacing of 40mm. for wing faces and wheel and Tetrahedral-T-grid with 60mm. of spacing for the volume. The mesh spacing was kept large to avoid excessive time consumption in computing.
The boundary conditions have been set as follow: [6]
-the inlet of the domain:
The flow upstream of the front wing is not disrupted by the other devices of the car even if the flow may be disrupted if the car follows another one.
So the inlet was set as "*velocity inlet*" with an intensity and length scale turbulence type:
-the airspeed at the entrance is equal to 60 m/s.
the turbulence intensity is set to 3% which is a standard value recommended (to get suitable results for the wheel & other turbulence intensity (down to 0.1%) coefficients were tried but the results did not change as expected.
-the length scale is set to 0.3 m which is the average length of the chord of the two airfoils from
-It is obvious for the side that deals with the centre of the car as the latter was split into two sections.

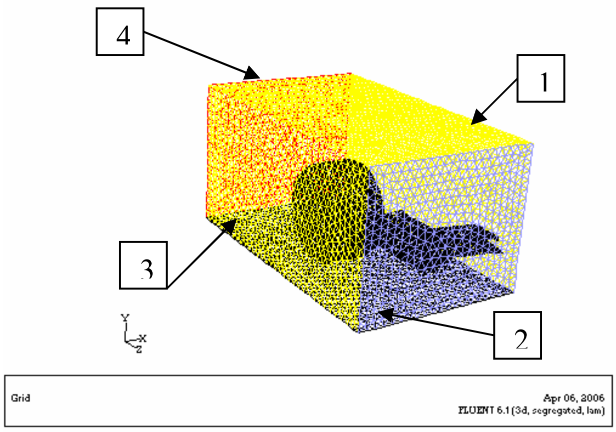

Figure 2.
1-symmetry 2-Velocity Inlet 3-Wall 4-Outflow

Table 3
Boundary Conditions

S. no	The part	Boundary	Type of motion	Speed

.		condition		
1.	Front wing	Stationary wall	None	NA
2.	Road	Moving wall	Translational	60m\s
3.	Wheel	Moving wall	Rotational (about it's axis)	181.82rad\s
4.	Domain sides	Outflow	NA	NA
5.	Car centre side	Symmetry	NA	NA

As is clear from the boundary conditions, the real problem was analyzed with the application of relative motion. The moving objects in the actual problem were set stationary and the stationary objects were made moving wall (road and fluid). The basic idea for setting the problem like that lies in the fact that the relative motion between the two remains the same as in the actual problem. In wind tunnel testing, they use the same approach as defined for the problem out here with the air being set at a high velocity.

3.3 CFD SOLVER:-

The CFD solutions were carried out with Fluent 6.1. The latter is a solver of the Navier-Stokes equations. The remaining conditions for solving the problem were defined in the solver.

The grid size was checked and the 811 faces imported into FLUENT were smoothened and swapped for covering the entire face and volume. The transient component of the equation above was solved using the explicit approach. A segregated solver is used in such a method as the equations are only solved as dependents of time. The convergence criterion defined was of the order of 1e-03. For the convection component, the SIMPLE algorithm was used for restricting the computation time.

The file imported in the solver had 17760 nodes, 1530 mixed wall faces in zone 3, 2034 mixed wall faces in zone 4, 1472 mixed wall faces in zone 5, 770 mixed outflow faces in zone 6, 768 mixed velocity-inlet faces in zone 7, 4012 mixed symmetry faces in zone 8, 174173 mixed interior faces in zone 10, 89733 tetrahedral cells in zone 2.

The Reynold's number for this flow was calculated to be between 10^6 and $3x10^6$ [4]. So, the turbulent 'k-ε' model is selected as per the real conditions as it is a three dimensional model and the flow can be considered to a low Reynold's number flow. The CFD equations for the model have been shown in the review. The fluid selected was air naturally, and was set at a velocity of 60m/s in the direction opposite to the direction of the road. The convergence criterion and other force monitors were all defined with a convergence criterion of 1e-03. The solution for all the angles converged at 170 iterations.

4. Result:-

Table 4

Angle of attack	Co-efficient of lift(Cl)	Co-efficient of drag(Cd)	Ratio= (Cl/Cd)
- 1 degree	- 1.09	0.390	- 2.79
0 degree	- 1.14	0.400	- 2.85
1 degree	- 1.21	0.412	- 2.93
3 degree	- 1.25	0.423	- 2.95
4 degree	- 1.30	0.438	- 2.97
5 degree	- 1.27	0.454	- 2.79

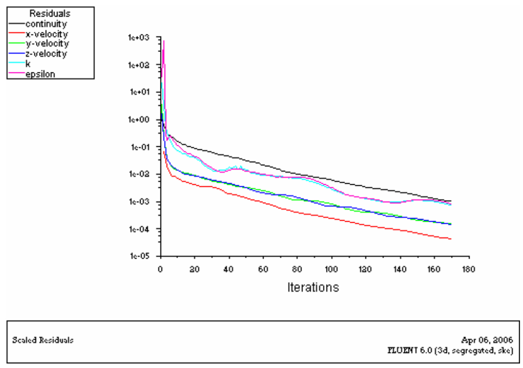

Figure 3. (Residuals plot)

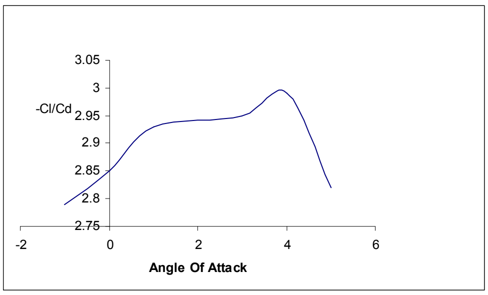

Fig. 4 (-C_L/ C_D v/s Angle of Attack)

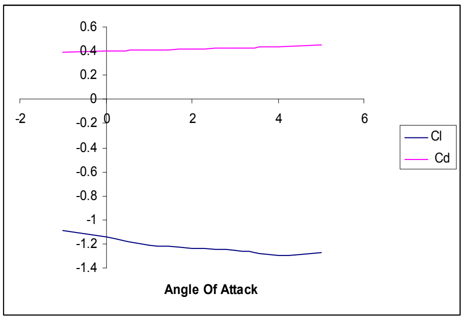

Fig. 5 (C_L & C_D v/s Angle of Attack)

5. Conclusion:-

From fig. 4, it can be inferred that the optimum angle occurs at 4degree. The downforce initially increases with the increase in angle of inclination but declines beyond four degree. The air is deflected by the air over the body of the car. The initial increase arises due to the deflected air having the effect on the flow of air over the body as it is deflected in close vicinity of the car body, but as the angle of inclination increases the air is deflected farther away from the body thus reducing it's influence on the airflow, and hence decreasing downforce.

The drag co-efficient rises as the angle of inclination is increased. This is mainly due to increase in the frontal area being exposed to the incoming air, thus providing more resistance to the airflow. As the angle of inclination increases, the exposed frontal area keeps increasing further.

The Monza circuit has long straight stretches forming significant portions of the circuit. To achieve top straight line speed in this portion, the designers should consider for the angle with the least value of C_D. So an angle less than the optimum angle of attack is to be considered. Monaco circuit, due to a number of high speed corners more downforce is significant than drag as there is not a straight stretch sufficient enough for the cars to reach top straight line speed. Better result can be achieved only with high speeds at corners in this circuit. The optimum angle of attack provides the highest value of C_l. So the front wing should be set at that angle. The Istanbul circuit has a good combination of straight stretches and turns. An angle between the optimum and with the least value of C_D can be decided upon depending upon the weather conditions, wind speed and other factors for that day. [15]

References:

1. **http://en.wikipedia.org\wiki\formula_one_cars-aerodynamics**
2. **Cirlunick Daniel, "Design considerations for Formula one", Engineer's Forum, pp 6- 7, September 2005 *issue*, Baker corporations (2005)**
3. **Hucho, Wolf-Heinrich" Aerodynamics of Road Vehicles, 4th edition", Pg. no. 396, SAE International,(1998)**
4. **F1 Racing, April 2005 *issue*, Pg. no. 61Haymarket Magazines Ltd., (2005)**
5. **Terry,L.N., "Extension of the Aerodynamic Design Program MSES for Simulating the Boundary Layer Suction", pp 29,(2004)**
6. **F Mortel, "Cranfield Team F1: The Front Wing",pp 12, 27, 31,www.public.cranfield.ac.uk/me/me028/rubini/thesis/2003/F.Mortel.2003.pdf (2003)**
7. **Anderson John D.,JR., "Introduction to Flight", pp. 239,McGraw-Hill Publications (2000)**
8. **www.cd-adapco.com/press_room/archive_2003/aerodynamic_simulation.html**
9. **Peter V. Nielsen, "The Selection of Turbulence Model for Prediction of Airflow", "International Journal of Building & Environment", pp.1129, ASHRAE Transactions: Symposia, Elsewhere Publications,**
10. **Emmerich Steven J., McGrattan Kevin B., "Application of a Large Eddy Simulation Model to Study Room Airflow", pp. 1129,1131, ASHRAE Transactions: Symposia, Elsewhere Publications,**
11. **Memarzadeh Farhad, Manning Andrew P., "Comparison of the Operating Room Ventilation Systems in the Protection of the Surgical Site", pp. 5, "International Journal of Building & Environment", ASHRAE Transactions: Research, Elsewhere Publications,**
12. **www.FIA.com**
13. **"Aerodynamics Features of the F1 Vehicle", 'Front wing and Nose Assembly', www.f1-country.com/f1-engineer/aerodynamics/f1-aerodynamics.html**
14. **www.gpracing.net192.com/cars/setup.cfm**
15. **F1 Racing, October 2005 *issue*, Pg. no. 144,152,160,Haymarket Magazines Ltd., (2005)**