CFD Analysis of Front Automotive Aerodynamic Airfoils

HAMDAN YOUSIF HAMDAN
Project and Construction Department, Mustansiriyah University
hamdanyousif2022@gmail.com

Abstract:- The aerodynamic profile of a car is altered through automotive aerodynamic technology. High-performance motorsport is one of the most competitive sports. In this study, computational fluid dynamics is used to analyze a Formula One front wing in order to better understand air flow characteristics and improve front wing design using airfoils, down-force, and drag scenarios. In this study, the experimental strategy is to add a NACA 23112 airfoil section to the front of the Formula One car. Simulating the selected airfoils based on angle of attack alterations correlated with two velocity values (30 m/s and 60 m/s) was used in the technique investigation. The airfoil behavior was examined and observed in terms of down force and drag. The ideal angle rang in all cases was 7.60, according to the experimental results.

Keywords:- aerodynamic forces, airfoil, lift force, drag force, airfoil, car wing, formula 1.

INTRODUCTION

For decades, aerodynamic properties have been a burgeoning and fruitful field of study. It's a branch of fluid dynamics that studies how air moves, especially when it collides with a solid object. The complex aerodynamics of automation is one of the primary research subjects. Its concerns in automobile design include lowering drag and wind noise, minimizing noise emissions, and avoiding unwanted lift forces and other causes of aerodynamic instability at high speeds. It also concerns producing down-force to boost traction and hence cornering abilities in racing automobiles. [1].
The Formula One automobile is a one-of-a-kind global platform for Formula One promotion. [2]. The goal of the Formula One in School Technology Challenge is to provide students with hands-on exposure with cutting-edge manufacturing and design technology, such as CAD/CAM, CNC, and CFD software. The evaluated automobile sketched geometry must be near to the production shape [3]. The necessary simplifications during the CFD treatment should be as minimal as possible. Due to their intricacy, surface-based CAD-systems are primarily used to create them. [4]. A consistent definition of completely linked geometry must be ensured during the preparation for a flow simulation. This first stage involves cleaning up the CAD model and is completely independent of the simulation technology used later. [5].

BACK GROUND OF STUDY

Aerodynamic study using Computational Fluid Dynamics (CFD) is a useful design tool. It assesses the viability of a given design using big object finite element models for later CFD analysis. [6]. Vehicle aerodynamic design has recently been increasingly essential to automobile manufacturers, as creating automobiles with lower drag and hence lower fuel consumption has become a key selling factor. [7]. In addition, aerodynamics has become one of the most important aspects of modern racing cars, and seemingly minor adjustments in aerodynamic setup can result in significant lap-time differences. From this perspective, our research will focus on Formula One racing cars, namely the front end wing, which is the most important component. [8]. The design condition of airfoils is heavily influenced by the surroundings of the front end wing. Improve cooling intake/exit and underbody flow by generating front load and reducing front bodywork lift and front wheel wake impact. [9]. The shape of the wing is concerned with the airfoil of Fl wings. The aerodynamic force is created by the movement of fluid on the form body. The lift force is perpendicular to the motion direction, while the drag force is parallel to the motion direction. Front wings, end plates, and a front Gurney Flap are the front end devices on this car. The race car tires sustain the vehicle's mass and transmit down force to the road. Low pressure must pull the car back to allow it to move around. [10]. The airflow surrounding the car is affected by the front wing, and the shapes of the wing are used to generate down force. The position of aerodynamic ‘wings’ is limited to the front wing's vertical envelope. [11].

RESEARCH METHODOLOGY

ANSYS Fluent is the most sophisticated computational fluid dynamics (CFD) software product on the market, allowing researchers to go further and quicker in their research. Fluent comes with well-validated physical modeling capabilities that give rapid, accurate results in a wide range of CFD and multiphysics applications. It also provides the wide physical modeling capabilities required to simulate turbulent flow, including multiphase models. CFD provides insight into flow patterns that would be difficult, costly, or impossible to analyze using traditional (experimental) methods. In most cases, CFD does not totally replace measurements, but it can greatly minimize the quantity of testing and overall expense. It is a better way to study the suggested product features rather than testing the new design or product.

Table 1: CFD Analysis Process

	Steps	Process
1	Problem Statement	Information About Flow
2	Mathematical Model	Generate 3d Model
3	Mesh Generation	Nodes/Cells, Time, Instants
4	Space Discretization	Coupled System
5	Time Discretization	Algebraic System Ax=B
6	Iterative Solver	Discrete Function Values
7	CFD Software	Implementation, Debugging
8	Simulation Run	Parameters, Stopping Criteria
9	Post Processing	Visualization, Analysis Of Data
10	Saving Case And Data	Save All Obtained Data

RESULTS AND DISCUSSION

The most crucial issue in formula one car design is aerodynamics. It is primarily concerned with producing the least amount of drag force in order to avoid slowing the automobile down, as well as creating the most amount of down force in order to improve cornering forces. This study presents several simulation tests of two types of wings using different angle of attacks in order to understand the behavior of these two forces based on distinct airfoil design. The tests are meant to use a variety of air velocity values to simulate real-world formula one speed situations.

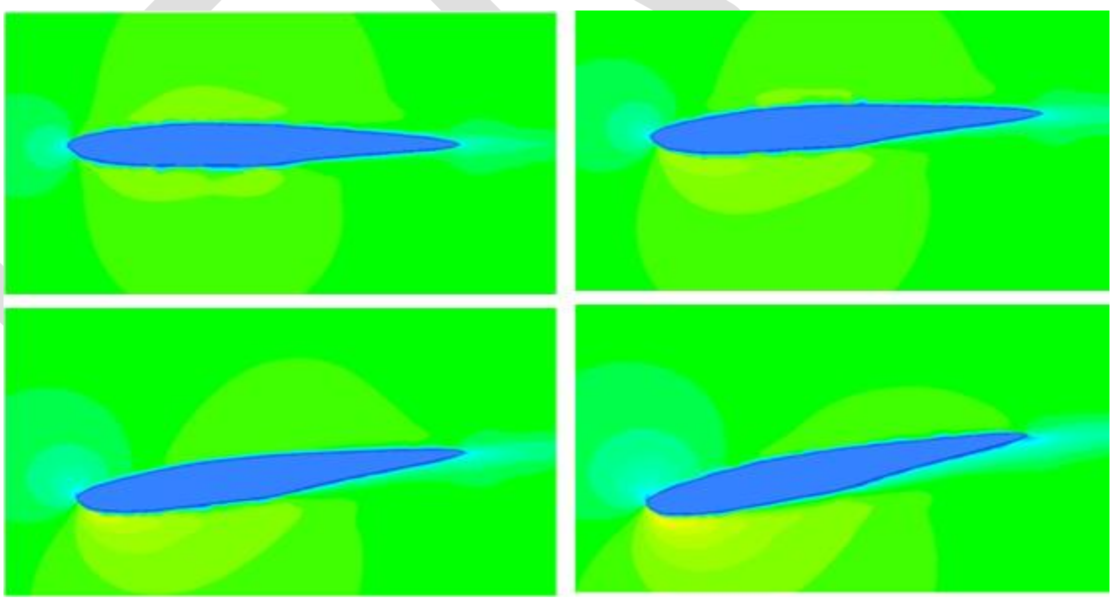

Figure 1: velocity profile of NACA 23112 with different angle of attack

The experiments using NACA 23112 were done using NSYS CFD Fluent software at 60 m/s and 30 m/s speeds. For the airfoil velocity and pressure, the Reynolds- Nayier-Stokes equations were solved using ANSYS CFD software. The flow behavior of the turbulence model was determined using numerical simulation and is presented in the sections below. The velocity and pressure contours for each run were calculated using simulation. When the flow velocity is set to 30 m/s, the first situation occurs. The velocity distribution over the wing is depicted in Figure 1. The figure depicts the variance of velocity flow in this velocity value. The tip of the wing has a lower velocity than the upper and lower surfaces. The flow velocity at the top and bottom of the wing is greater than the airfoil's boundary condition, particularly at the bottom of the wing, where the flow is greatest in this example, as shown in red. The pressure distribution in Figure 2 confirms this phenomenon. The pressure contour is shown in blue, indicating the lowest pressure region. The wing front surface is the principal portion exposed to the lowest pressure. This is because of the airfoil form of this surface. The shape of the airfoil is important because it regulates air movement, which modifies the surface pressure. The parameter that causes the force effect on the wing is pressure.

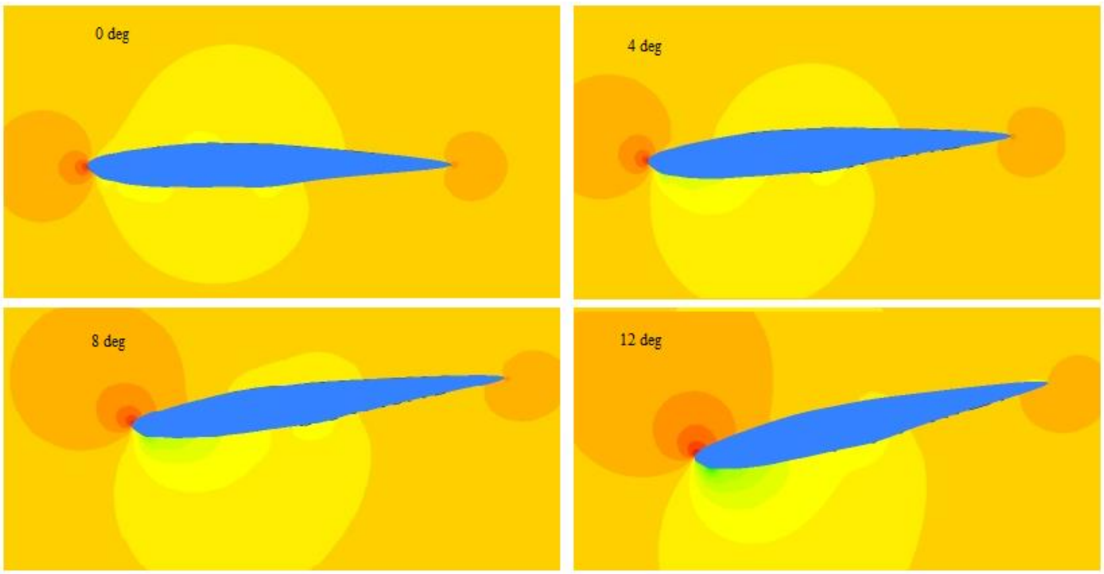

Figure 2: pressure profile NACA 23112 with different angle of attack

In this situation, the inflow forces below the wing and the outflow forces above the wing both alter in a positive way. These modifications aid in the normalization of down force distribution across the entire wing surface. This occurrence is the first step toward boosting down force and lowering drag.

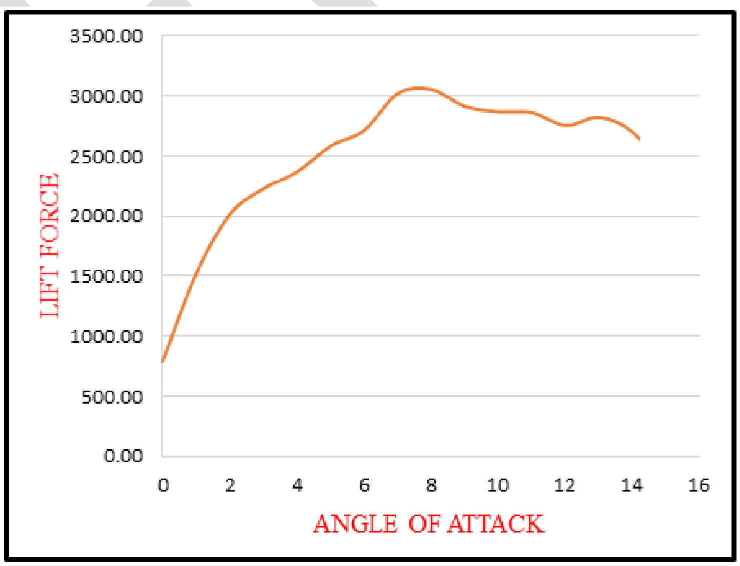

Figure 3: correlation between the angle of attack and the lift force of NACA 23112

The second set of tests and findings show the same pattern in terms of air flow and pressure effect. Different amounts of lift force, as well as lift and drag coefficients, are produced by variances in numerical properties depending on different angles of attack. The aerodynamic impact due to lift force created by air speed of the NACA 23112 wing section is included in the CFD findings of this model. This scenario was used with a variety of attack angles. The lift force increases as the angle of attack increases, according to the findings. Figure 3 shows the relationship between the lift force and the angle of attack. The data show that the lift force increases as the wing angle of attack increases until it reaches 7.60, at which point it decreases. The lift force increases as the angle of attack increases, according to the findings. The lift coefficient values increase at 7.60 and thereafter decline, according to the data. Based on the same phenomenon, the growth in lift coefficient values decreases beyond the 7.60 angle, even though the air velocity increases. Drag is caused by variations in velocities between the wing object and the air. The difference is due to the drag of the NACA 23112 wing section, which causes the angle of attack to alter. The data also show that the drag coefficient rises as the air speed rises.

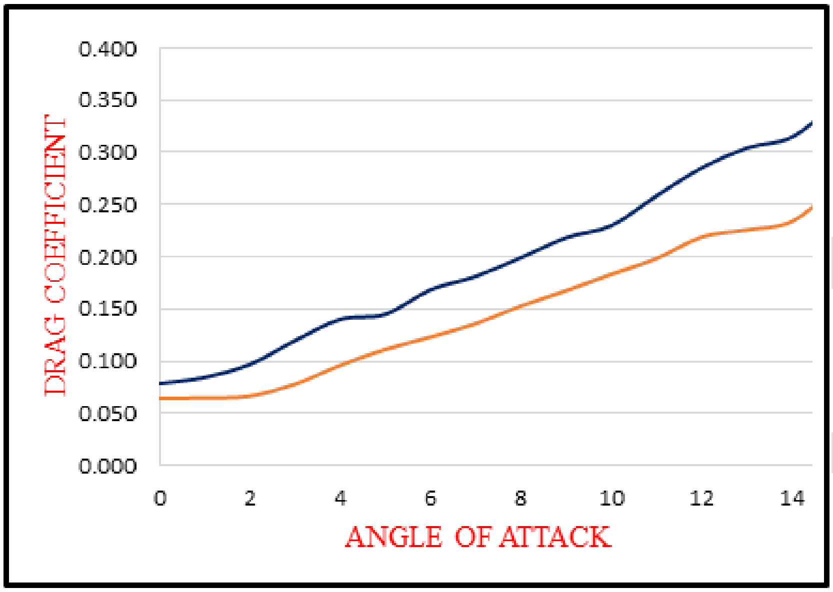

Figure 4: the correlation between the angle of attack and the drag of NACA 23112 wing section

The stall angle of attack is the crucial angle at which the highest lift coefficient is produced. In the case of the NACA 23112 stall angle, it is obvious that the angle 7.60 is the stall. Its purpose is usually to raise the maximum lift coefficient. To increase the critical angle of attack (stall angle) at the wing tips, an aircraft designer may lessen the camber of the outboard section of the wings. When the wing approaches the stall angle, the wing root stalls before the tip, preventing the airplane from spinning and preserving aileron efficacy near to the stall.

CONCLUSIONS

The study finds that the results of different parameters, adding knowledge regarding the control of the airfoil streamline, which allows dominating the flow stream and pressure to help the forces for the Front tires. The study finds that the wing design is concerned with the task's goal. To reduce drag, the top speed goal necessitates lowering the angle of attack. In order to avoid slipping, the angle of attack must be raised to maximize down force, which increases friction between the front ties and the ground. The performance results of NACA 23112's CFD findings revealed a crucial deduction, as shown below:

a) CFD findings showed an increase in the generated down force from the air effect on the wing in the 30 m/s scenario.

b) In the 60 m/s velocity scenario, CFD findings revealed an increase in the generated down force due to the air effect on the wing. In this test speed, the observation provides an essential viewpoint; it shows a drop in value over the 90 angle. These findings suggest that at all angles and velocity instances, the down force magnitude does not behave symmetrically. The amount of the down force fluctuation provides the best wing angle range values. In genuine formula one design, this information allows the wing designer to build and select the ideal wing angle based on the required down force.

ACKNOWLEDGMENT

The author wants to acknowledge Project and Construction Department, Mustansiriyah University to support this work.

REFERENCES:

[1] Mokhtar, W. and Durrer, S., 2016. A CFD Analysis of a Race Car Front Wing in Ground Effect. Book

[2] Raul, V.V., 2013. *Analysis of F-duct drag reduction system in Formula 1*(**Doctoral dissertation**, Wichita State University).

[3] Will Tyson 2014. Front F1 aerodynamics .www.WTPformula.co.uk Accessed at 6th November 2015..

[4] Adel Muhsin Elewe. "Numerical simulation of surface curvature effect on aerodynamic performance of different types of airfoils". IOP Conf. Series: Materials Science and Engineering 928 (2020) 032003 IOP Publishing doi:10.1088/1757-899X/928/3/032003.

[5] Patil, A., Kshirsagar, S. and Parge, T., 2014. Study Of Front Wing Of Formula One Car Using Computational Fluid Dynamics. International Journal Of Mechanical Engineering And Robotics Research, 3(4), P.282.

[6] Tritthart, M., Mayrhofer, A., Glas, M., Glock, K 2015. and Habersack, H., Comparison Of The K-Epsilon And K-Omega Turbulence Models In A Laboratory And A River Modelling Context.

[7] Vadgama, T.N., Patel, M.A. and Thakkar, D., 2015, April. Design of Formula One Racing Car. In *International Journal of Engineering Research and Technology* (Vol. 4, No. 04 (April-2015)). ESRSA Publications

[8] Will Tyson 2014. Front F1 aerodynamics .www.WTPformula.co.uk Accessed at 6th November 2015.

[9] Zhang, Y., Gillebaart, T., van Bussel, G.J.W. and Bijl, H., 2013. Validation of a transition model for the DU91-W2-250 airfoil. In Proceeding of 9th PhD seminar on wind energy in Europe, Sweden.

[10] Zhang, X., Toet, W., Zerihan J., 2006, "Ground Effect Aerodynamics of Race Cars, ASME Appl. Mech. Rev.," 59 (1-6), pp. 33-48. DOI: 10.1115/1.2110263

[11] Wordley, S. and Saunders, J., 2006. Aerodynamics for Formula SAE: Initial design and performance prediction (No. 2006-01-0806). SAE Technical Paper.