

## Aerodynamics of a Formula One Car Front Cascade Wing during Cornering

Open  
Access

Ahmed T. Raheem<sup>1</sup>, Azwan Sapit<sup>1,\*</sup>, Akmal Nizam Mohammed<sup>2</sup>

<sup>1</sup> Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

<sup>2</sup> Centre for Energy and Industrial Environment Studies, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

### ARTICLE INFO

### ABSTRACT

#### Article history:

Received 10 October 2018

Received in revised form 21 November 2018

Accepted 5 December 2018

Available online 8 January 2019

The design for the aerodynamics of front wing based on cornering angle of a Formula-One car plays an important role on the car's performance. The cascade elements above the main front wing is enhanced with new and the deep winglets that manage the airflow for the rest of the car. The front wing cascades are attached with end plates at the extremities of the front wing to reduce turbulence. However, when the car turns, the airflow behaviour on these cascades wings changes significantly. This paper presents the aerodynamic characteristics resulting from the cornering forces subjected on Formula One styled cascade wings. The study attempts to predict the down-force and drag-force acting on the front wing under the effect of airflow change in this area. In order to investigate the airflow behaviour, computational fluid dynamics (specifically the ANSYS software) was used to simulate selected cases with specific surface definitions and boundary conditions defined in a 3D domain. It can be observed that with the increase in the car's yaw angle during turning, the down force and the drag force decrease in an inversely proportional manner.

Aerodynamics, front cascade wing,  
down-force, drag-force, CFD analysis

Copyright © 2019 PENERBIT AKADEMIA BARU - All rights reserved

## 1. Introduction

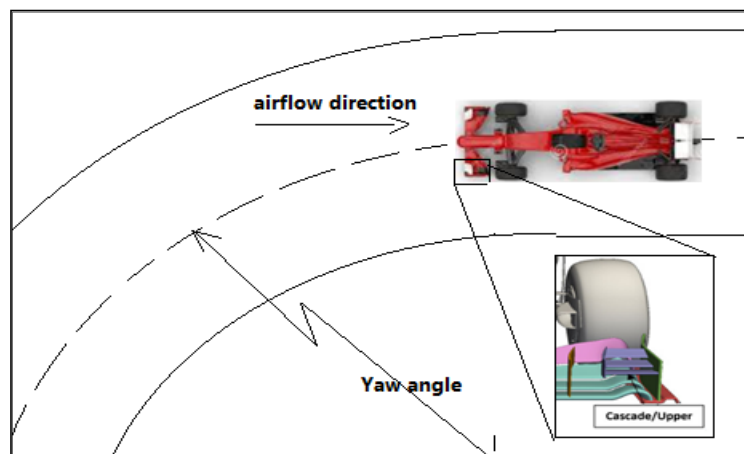
One of the most important and significant aspect in Formula One design is the field of aerodynamics. The aerodynamic designers have two main concerns: Generation of down force in order to stabilize the car during turning and to improve cornering speeds; and minimizing the drag force, caused by turbulence, which in turn reduces the speed of car [1,2]. Therefore, improvement in these factors could improve the overall performance of the car; the front wing shape and position have a major effect on the center of pressure and the down-force produced by the flow stream. The front wing of a Formula One car creates about 25% of the total car's down-force. Designs such as in [3, 4] are one of the most widely used spoiler airfoil, but needs to be enhanced as per requirements related to running speeds. Therefore, many researchers try to develop this part of the car. In 1997, the Minardi M197 introduced the curved front wing in Formula One car. McLaren F1 Team developed the front wing by adding multi-elements to the end plates. In 2009, the cascade elements have been

\* Corresponding author.

E-mail address: [azwans@uthm.edu.my](mailto:azwans@uthm.edu.my) (Azwan Sapit)

used by introducing small wings above the main board [5, 6]. Their purpose is to provide more to control the airflow around the front wheels, managing flow regimes for the rest of the car and generating down-force [1]. Keogh *et al.*, [7] in 2014 studied the effects of front wing based on cornering condition based on a preceding study in 2011. They studied the airflow in the front wing due to the yaw angle. The study observes significant results of flow curvature, a velocity gradient and yaw. These results illustrated the pressure distribution over the wing surface and the path of the prominent vortices generated at the end of the span [8]. Based on the present studies, this paper will discuss the effect of cascade wings in cornering condition as shown in Figure 1.

The aim of the study is to investigate the effect of these elements on a Formula One car's performance. The cascade wings on a racing-car are small elements pushing the airflow over and around the wheels in order to increasing the performance of the components downstream. The aerodynamic performance is considered the most important in corners. Generally, designs will typically be evaluated through straight-line testing. There is awareness within industry of the difference in the conditions experienced through a corner. Therefore, this work studies the differences in aerodynamic performance of front cascade wings through a radius corner based on the yaw angle.



**Fig. 1.** Front cascade wings through radius corner

## 2. Front Wing Aerodynamics

Aerodynamics is a branch of the study of flow that concentrates on the motion of air, specifically when it interacts with solid object [9]. The aerodynamic force is the relative motion between the body and the air. Aerodynamic force is the normal force caused by the pressure on the surface of the body and shear force due to the viscosity of the gas, which is also known as skin friction. Pressure acts locally, normal to the surface, and shear force acts locally, parallel to the surface. The aerodynamic force of the net on the body is due to the pressure and shear forces integrated over the total exposed area of the body. This aerodynamic force is usually resolved into two components [10]; the first is the drag force component parallel to the direction of relative motion and the second is the lift force component perpendicular to the direction of relative motion. In addition to these two forces, the body may experience an aerodynamic moment also, the value of which depends on the point chosen for calculation. The aerodynamic design of a racing car is performed to achieve the best compromise between low drag and high inverse lift or down-force [2,3]. The aerodynamic drag is the drag force caused by the air through which the car is moving. It consists of pressure and viscous drag.

The air Drag Coefficient is a constant figure for a particular object. It is a measure of how good the aerodynamic shape of the object is. To get a lower drag you need a lower frontal area and/or a lower air drag coefficient. The drag performance of vehicles is characterized by the drag coefficient (CD) which is defined as studies reported by Devaiah and Umesh [5].

$$C_D = \frac{F_D}{\left(\frac{1}{2}\right)\rho V^2 A} \quad (1)$$

where  $F_D$ ,  $\rho$ ,  $V$  and  $A$  are the drag force, the density of air, the free stream velocity and the frontal area of the vehicle respectively. This dimensionless coefficient makes it possible to compare the drag performance between different vehicles directly with different preparations for the same vehicle. The lift is the other of the two major aerodynamic forces imposed on race cars, but unlike the drag, the lift can be manipulated to improve the performance of the race car and reduce lap times [11, 12]. Lift is a force acting on a normal vehicle to the surface of the road that the vehicle is riding. As meant by definition, lifting usually has the "pull" effect of the vehicle from the driven surface. However, by manipulating the geometry of the racecar, it is possible to generate a negative lift force or down-force. By increasing the normal load on the tires, more down-force is produced to improve the vehicle performance [13]. This increases the potential cornering force, which results in the ability of the vehicle to corner faster and reduce lap times. The lift of the vehicle is characterized by the lift coefficient ( $C_L$ ) and is defined as reported by Devaiah and Umesh [5].

$$C_L = \frac{F_L}{\left(\frac{1}{2}\right)\rho V^2 A} \quad (2)$$

where  $F_L$  is the lift force,  $A$  is the area of the upper surface of the vehicle, and the other variables are as defined above. A negative lift coefficient means that a vehicle is experiencing down force [14]. The performance of a racecar can be greatly enhanced by the production of down-force. As such, the main design objective for a front wing is to generate as much down-force as possible [14]. Based on the present review, this paper will study the effect of cascade as a part of front wing in formula one car in case of cornering. The importance of this case is to know how the air behavior in this part in each angle of turn.

### 3. Wing Simulation

#### 3.1 Validation

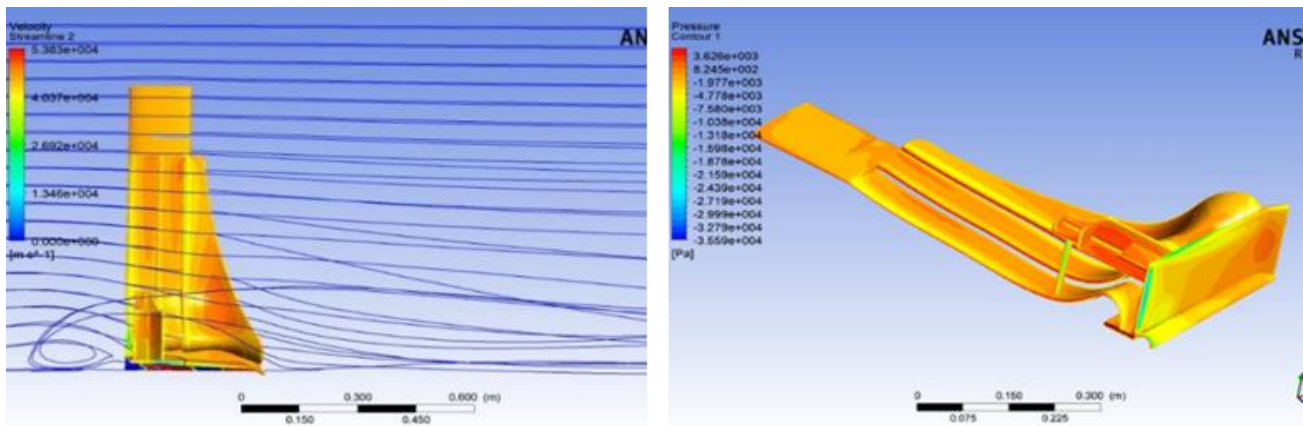
Model of the front wing was designed in the 'SOLIDWORKS 2016' and considering all the above-mentioned factors influencing the aerodynamic properties. In order to investigate the airflow behaviour, the Computational fluid dynamics able to analyses problems that involve fluid flows. The ANSYS software will simulate the selected cases with surfaces definition by boundary conditions. There are several fundamental properties used in the CFD-ANSYS simulation process, as in Table 1.

The car front wing is symmetric about a central vertical plane, thus only half portion is used for analysis. Mesh model selected as tetrahedrons with minimum element size =  $(6 \times 10^{-5})$  m and average skewness = 0.2.

**Table 1**  
 Operating Parameters

Item	Parameter	Value
1	Fluid Type	Air
2	Reynolds number, Re	$4.6 \times 10^5$
3	Operating temperature	288.16 K
4	Operating pressure	101325 Pa
5	Turbulence model	SST k- $\omega$
6	Fluid Density	1.225 Kg/m <sup>3</sup>
7	Viscosity	$1.7894 \times 10^{-5}$ kg/m-s
8	Yawing Angles	0 to 25

Several scenarios are generated using the behaviour of the pressure and velocity in various yaw angles. In order to generate the scenarios, the yaw angles are identified in zero angles. The analysis is based on the flow around the model using software FLUENT similar to the study in [15]. The values of  $C_L$  and  $C_D$  for different yaw angles have been obtained from FLUENT after respective analysis. Convergence of solution for all models is obtained after about 200 iterations. Figure 2 observes the simulation results for velocity and pressure profiles:



**Fig. 2.** Simulation results for velocity and pressure profiles

The CFD simulation model has been valid and compare with James Keogh, who prepare their paper in 2015. The  $C_L$  and  $C_D$  changes for conditions of  $Re = 4.6 \times 10^5$  and a zero yaw angle are presented in Table 2.

**Table 2**  
 $C_L$ ,  $C_D$  Numerical Settings for a zero yaw angle

Coefficient	Previous Study	This Study
$C_L$	1.28	1.245
$C_D$	0.055	0.0571

The significant results pave the way to study the cascade element in order to investigate the  $C_L$  and  $C_D$  changes due to different yaw angles.

### 3.2 Cascade Wing Results

The cascade wings consist of two elements of air foils connected to each other by the wing endplates. It has provided good protection for the body from the direct air streamline. The Reynolds-

Navier-Stokes equations have been solved by ANSYS CFD software for the air foil velocity and the pressure. The turbulence model flow behaviour was obtained from numerical simulation are presented based on different yaw angles. The contours from each run were obtained from simulation for the velocity and pressure.

The first scenario is for a zero yaw angle. Figure 3 observe the velocity and pressure profile of different yaw angles. The figures show the velocity behavior around the wing. The velocity figure observes the variation of velocity flow behind wing. The velocity in the tip of the first wing is higher than upper and bottom surface of the wing with low effect area. The pressure distribution shows that the lowest pressure region in blue color at the top of second wing. While the pressure concentrated in the first wing. The pressure is the parameter that produces the force effect on the wing. It is seen also that the upper wing resist the airflow. This resistance can be used as a positive phenomenon by changing the air stream out of the car body. However, it also considered as a negative force based on the position of the pressure concentration on the wing. It develops the drag force.

In order to understand the airflow behavior, the yaw angle has been changed. The numerical results of aerodynamic performance characteristics differences in aerodynamic characteristics based on different yaw wing angle values for down force, drag, lift coefficient and drag coefficient presented in Table 3:

**Table 3**  
Results on down force and drag force, also  $C_L$  and  $C_D$  for each yaw angle

Yaw Angle	Down force (N)	Drag force (N)	$C_L$	$C_D$
0	39.7	5.641	1.245	0.0571
5	36.2	3.93	1.2325	0.0571
10	34.3	3.449	1.19225	0.0569
15	27.9	2.565	1.18225	0.0569
20	22.8	2.324	1.1545	0.05685
25	11.4	2.13	1.1265	0.0568

Figure 3 also observes the result of 0, 10 and 20 degree of velocity and pressure yaw angle profile. The down force in the table observes an expected decrease in the generated force from the air effect on the cascade wing.

The turbulence of flow in the end sides of the wing increases due to the increase of yaw angle. This phenomenon approved by the pressure distribution in Figure 3. It shows that the pressure contour observes the highest-pressure region in red color in the wing top surface. The pressure is the parameter that produces the force effect on the wing. Therefore, this may cause high friction loss between the tires and the ground. The friction in this case can prevent the slip when the car turns. But with higher pressure, it slows down the car speed. Figure 3 shows a clear increase in the turbulence behind the wing end. That means a possible increase in friction value between the air and the tire surface.

It is clear from the results that the pressure value is less the zero yaw condition, and the airstream after the wings is in the lower level than in a zero yaw angle. The reason in decreasing the down force is due to the decreases of the pressure force in the first cascade wing surface, and the lift coefficient and drag coefficient values observe an expected decrease due to the increase in the yaw angle. The angles from zero to 20° observe a decrease in lift coefficient and drag coefficient especially in angle 10°. This angle can be considered the optimal angle due to the sudden drop in drag coefficient values, Figure 4 observe a comparison between these parameters.

#### 4. Conclusions

With respect to the CFD simulation work, the study observes that the cascade wings streamline allows better control to the flow stream and pressure to aid the forces for the front tires. The study observes that as the car's yaw angle increases during turning or cornering, the air stream around the front wing is affected in such a way that both the downforce and drag is reduced. Furthermore, the level of downforce and drag is inversely proportional to the severity of the cornering angle or the direction of oncoming wind as compared to the straight-line position. Consequently, the overall downforce acting on the car is reduced as well, meaning that the car's stability would be adversely affected when the turning angle becomes more acute. Therefore, further investigation should be made to develop better designs that would cater for both straight-line performance and cornering stability.

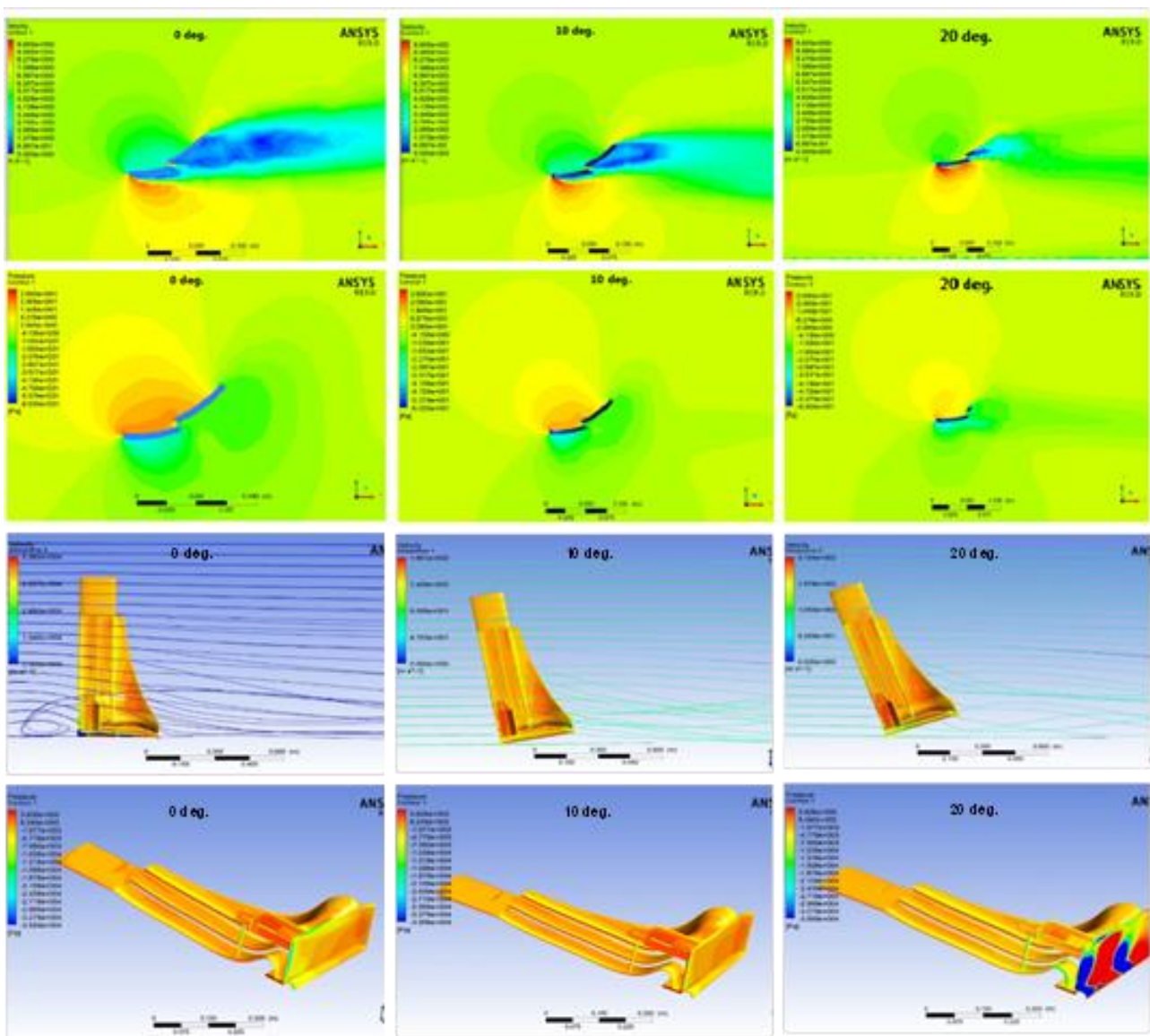


Fig. 3. Velocity and pressure profile of different yaw angle

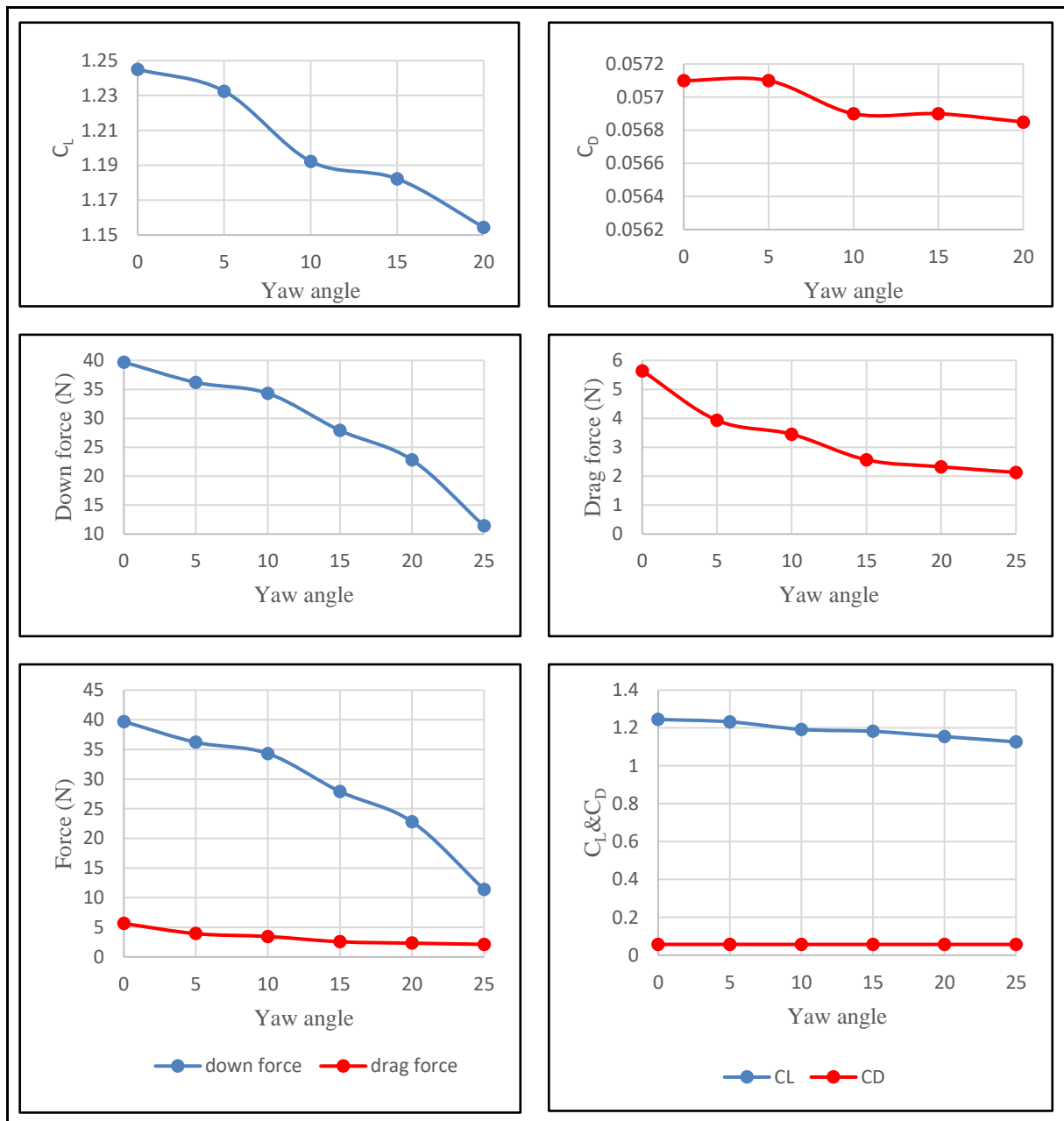


Fig. 4. Lift coefficients, drag coefficients, down force and drag force changes with yaw angle

### Acknowledgement

The authors would like to thank the Flow Analysis, Simulation and Turbulence Research Group (FASTREG) of the Centre for Energy and Industrial Environment Studies (CEIES), as well as Universiti Tun Hussein Onn Malaysia (UTHM) for their support of the present work through internal grant (IGSP Grant U413).

### References

- [1] Patil, Aniruddha, Siddharth Kshirsagar, and Tejas Parge. "Study of Front Wing of Formula One Car Using Computational Fluid Dynamics." *International Journal of Mechanical Engineering and Robotics Research* 3, no. 4 (2014): 282.
- [2] Toet, Willem. "Aerodynamics and aerodynamic research in Formula 1." *The Aeronautical Journal* 117, no. 1187 (2013): 1-26.
- [3] Vadgama, Triya Nanalal, Mr Arpit Patel, and Dr Dipali Thakkar. "Design of Formula One Racing Car." *International Journal of Engineering Research & Technology (ISSN: 2278-0181)* 4, no. 04 (2015): 702-712.

- [4] Diasinos, S., G. Doig, and T. J. Barber. "On the interaction of a racing car front wing and exposed wheel." *The Aeronautical Journal* 118, no. 1210 (2014): 1385-1407.
- [5] Devaiah, B. N., and S. Umesh. "Enhancement of Aerodynamic Performance of a Formula-1 Race Car Using Add-On Devices." *SASTech Journal* 12 (2013): 72-79.
- [6] FIA, 2009 Formula One Technical Regulations, Article 3 (Bodywork and Dimensions).
- [7] Doig, Graham, Tracie J. Barber, and Andrew J. Neely. "The influence of compressibility on the aerodynamics of an inverted wing in ground effect." *Journal of fluids engineering* 133, no. 6 (2011): 061102.
- [8] Keogh, James, Graham Doig, Tracie J. Barber, and Sammy Diasinos. "The aerodynamics of a cornering inverted wing in ground effect." *Applied Mechanics and Materials* 553 (2014): 205.
- [9] A. Maniyar, V. Sonawane, and N. Gorade, "Aerodynamic Study of F1 car," *Int. J. of Res. in Advent Tech.*, March special issue, pp. 15–18. 2017.
- [10] Flay, Richard GJ, Andrew R. Hammond, N. Zealand, and F. M. Student. "Aerodynamic design of a formula sae race car." In *BBAA VI Int. Colloq. Bluff Bodies Aerodyn. Appl*, pp. 1-4. 2008.
- [11] Gupta, Sahil, and Kishal Saxena. "Aerodynamics analysis of a formula sae car."
- [12] Schommer, Adriano, Paulo Soliman, Lucas Travi Farias, and Mario Martins. *Analysis of a Formula SAE Vehicle Suspension: Chassis Tuning*. No. 2015-36-0275. SAE Technical Paper, 2015.
- [13] Petkar, R. K., S. G. Kolgiri, and S. S. Ragit. "Study of Front-Body of Formula-One Car for Aerodynamics using CFD." *International Journal of Application or Innovation in Engineering & Management (IJAIEM)* 3, no. 3 (2014).
- [14] Pegrum, Jonathan Mark. "Experimental study of the vortex system generated by a Formula 1 front wing." PhD diss., Imperial College London, 2007.
- [15] Cheng, Xiao Han, Shan Ming Luo, Xue Feng Chang, and Dan Xie. "Numerical Analysis of an External Flow-Field around a Formula SAE Car Body based on FLUENT." In *Advanced Materials Research*, vol. 1039, pp. 17-24. Trans Tech Publications, 2014.