

# Aerodynamic Analysis of F1 IN SCHOOLS™ Car

S. J. Lim<sup>1</sup> and M. R. A. Mansor<sup>\*,1,2</sup>

<sup>1</sup>Department of Mechanical & Material Engineering, Faculty of Engineering & Built Environment, Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Malaysia

<sup>2</sup>Centre for Automotive Research, Faculty of Engineering & Built Environment, Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Malaysia

\*Corresponding author: radzi@ukm.edu.my

## ORIGINAL ARTICLE

Open Access

### Article History:

Received  
15 Dec 2016

Received in  
revised form  
10 Jan 2017

Accepted  
15 Jan 2017

Available online  
31 Jan 2017

**Abstract** – F1 IN SCHOOLS™ is a worldwide competition that is part of the efforts undertaken by the STEM educational model. In order to increase the performance of the F1 IN SCHOOLS™ car in terms of speed, two important parameters related to aerodynamic analysis are considered – drag coefficient and downforce coefficient. Drag force is a force that acts in the direction that is opposite of the car's motion, thus reducing the car's maximum speed. Meanwhile, sufficient downforce is beneficial to the car model because it allows the car's wheels to remain in contact with the track surface without going off-track. The most important component of a F1 IN SCHOOLS™ car is its front wing since its design has a significant effect on the drag coefficient and downforce coefficient induced by the air flow. Therefore, the objective of this study is to design a front wing that is capable of producing low drag coefficient while maintaining sufficient downforce coefficient. Moreover, this study also aims to examine the method of preventing flow separation at the rear part of the car model. This study will use Autodesk Inventor Professional to create the car model. The simulation will be run using the STAR CCM+ software. The simulation will also be used to obtain the drag coefficient and downforce coefficient of the car.

**Keywords:** F1 IN SCHOOLS™, aerodynamics, drag force, CFD, STEM

**Copyright** © 2017 Society of Automotive Engineers Malaysia - All rights reserved.

Journal homepage: [www.journal.saemalaysia.org.my](http://www.journal.saemalaysia.org.my)

## 1.0 INTRODUCTION

F1 IN SCHOOLS™ is a worldwide competition that aims to change the perspective of the student towards subjects such as science, technology, engineering and mathematics (STEM). This competition provides participants with a challenging and interesting environment that requires them to integrate and apply concepts such as aerodynamics, physics, fluid dynamics and mechanical design in order to create a quality F1 IN SCHOOLS™ car that will perform well in terms of speed and stability. The most important parameter that quantifies the performance of the F1 IN SCHOOLS™ car is its maximum speed. This maximum speed allows it to complete the 20-metre track within the shortest time possible. The maximum speed

attainable by the car model is significantly affected by its aerodynamics. In general, aerodynamics is a branch of fluid mechanics that focuses on the study of the behaviour of air flow and its interaction with the solid body within itself (McCabe, 2008). It enables this study to obtain the drag coefficient and downforce coefficient, which in turn will directly affect the maximum speed attainable by the F1 IN SCHOOLS™ car. Therefore, the primary concern of this study is the application of aerodynamics on the F1 IN SCHOOLS™ car.

The main purpose of applying aerodynamics during the design process of the F1 car is to obtain a F1 car with sufficient downforce and low drag force (Zhang et al., 2006). Drag force acts as a resistance force for an object accelerating in the air flow region (Hetawal et al., 2014). Drag force is a force that acts in a direction that is opposite of the car's motion. It is unavoidable and it creates a pulling force that slows down the car. Downforce is a reaction force acting on the car surface. It ensures that the car's wheel is always in contact with the track surface to prevent it from going off-track (Reddy & Mech, 2006). The main factors that contribute to the drag force on an F1 car are pressure difference in the front and rear parts of the car and flow separation zone at the rear part of the F1 car. When an F1 car operates at high speed, air molecules will tend to stagnate at the front tip of the car and increase stagnation pressure. This stagnation pressure will produce a force that acts in the opposite direction of the car's motion, and thus contributing to the drag force (Rakibul Hassan et al., 2014). Moreover, flow separation will contribute to drag force. Furthermore, its size will determine the magnitude of drag force on the F1 car (Guilmineau, 2008). The flow separation that takes place at the rear part of F1 car will create a low pressure zone that cannot be filled with air molecules. This leads to a large pressure difference between the front and rear part of the F1 car and forms a pressure drag. This pressure drag will slow down the speed of the F1 car since it induces a pulling force on the car's body (Rakibul Hassan et al., 2014). The result obtained from this study is the drag coefficient. This can then be used to calculate the drag force using Equation 1.

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

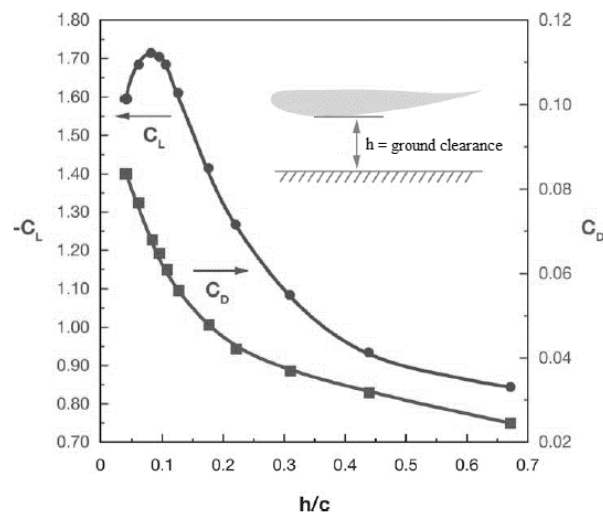
where  $F_D$  is the drag force,  $\rho$  is the density of air,  $V$  is the velocity of air and  $A$  is the frontal area of the object. Although drag force is unavoidable, it can be minimised by reducing air flow separation. This can be achieved by modifying the car body so that the air can be directed to flow into the flow separation zone. This can be seen in Figure 1(a) and Figure 1(b) below (Rakibul Hassan et al., 2014).



**Figure 1:** (a) Slicing of car body at an angle of  $\beta$  at the bottom rear part of the car model (b) Installation of diffuser at the bottom rear part of the car model (Rakibul Hassan et al., 2014)

Downforce plays an important role in maintaining the stability of the F1 car when it is operated at high speeds. Downforce on an F1 car can be generated through two approaches – ground effect of the F1 car and front wing installation. Ground effect refers to a phenomenon that increases the lift-drag ratio when a vehicle is located at a relative distance from the ground

surface (Rozhdestvensky, 2006) Applying ground effect on the front wing of the F1 car is vital because it is the first component that is exposed to the air flow. Thus, the air flow behaviour will start to change at this region, and the downforce will be affected (Soso & Wilson, 2006). Ground effect occurs when the ground clearance (distance between the bottom surface of the car and the road surface) decreases to an optimum height. This decrease is due to the fact that the airflow velocity accelerates when the ground clearance decreases. The acceleration of airflow results in high velocity airflow at the underbody of the car model. According to Bernoulli's principle, high velocity airflow will create a low pressure region. Therefore, the pressure at the underbody of the car model will be less than the pressure at the upper surface. This pressure difference will create a downforce that acts on the car body. However, when the ground clearance is reduced to a distance that is less than the optimum height, the high air flow velocity at the underbody will result into a greater flow separation and a reduced downforce (Kuya et al., 2009). Moreover, according to Figure 2, if the ground clearance is very low, the air at the underbody is assumed to be stationary. Hence, a high pressure region will be created at the car's underbody. At this point, a lift force will be created instead of a downforce.



**Figure 2:** Graph of  $C_L$  and  $C_D$  against  $h/c$  (Katz, 2006)

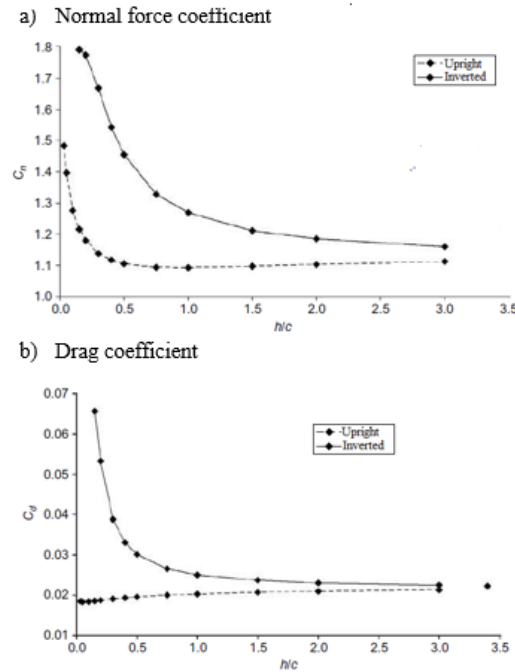
Although the ground effect can create a downforce on an F1 car, it also will bring along a significant drag force. This drag force will increase with the decrease in ground clearance as shown in Figure 2 (Katz, 2006). The downforce acting on the car body can be related to the downforce coefficient using Equation 2 as shown below.

$$C_{DN} = \frac{F_{DN}}{\frac{1}{2}\rho V^2 A} \quad (2)$$

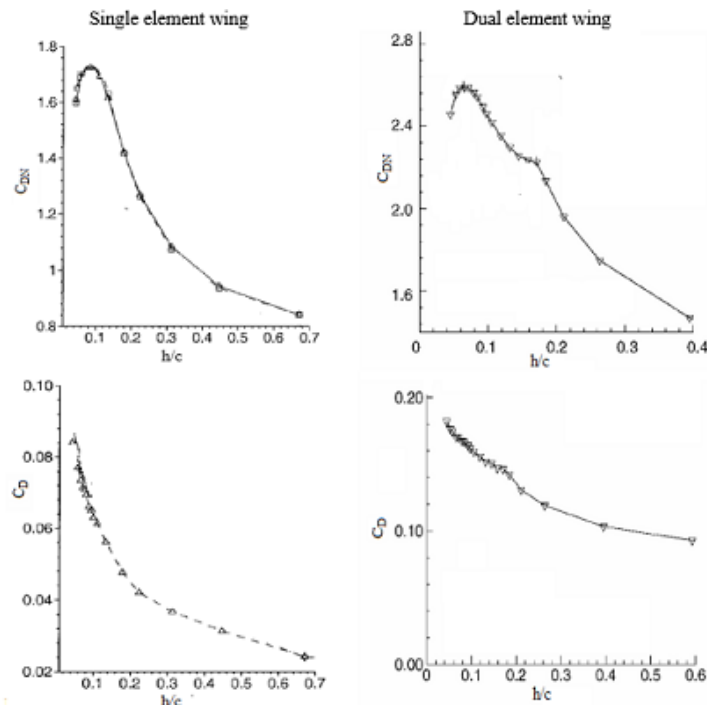
where  $C_{DN}$  is the downforce coefficient and  $F_{DN}$  is the downforce.

Another approach that contributes to the downforce is the installation of the front wing. Front wing design has two categories - the upright and inverted wings. The main difference between these two wings is the direction of the normal force created. The upright wing will create an upward normal force (lift force) while the inverted wing will create a downward normal force (downforce) (Vogt & Barber, 2012). According to Vogt and Barber (2012), the performance of upright and inverted wings in terms of creating downforce and drag force can be observed using Figure 3. As seen in Figure 3(a), the inverted wing is able to create a higher

normal force compared to the upright wing. Since the direction of the normal force produced by the inverted wing is downward, it implies that the inverted wing can create better downforce. However, the upright wing performs better in terms of creating drag force. This is due to the fact that it has lower drag coefficient compared to the inverted wing, as shown in Figure 3(b).



**Figure 3:** Graph of force coefficient against  $h/c$  for upright and inverted wing (Vogt & Barber, 2012)



**Figure 4:** Graph of force coefficient against  $h/c$  (Zhang & Zerihan, 2003; Zerihan & Zhang, 2000)

Furthermore, the number of wings in use will affect the drag force and the downforce created on the F1 car. Generally, single element and dual element wings are used in the front

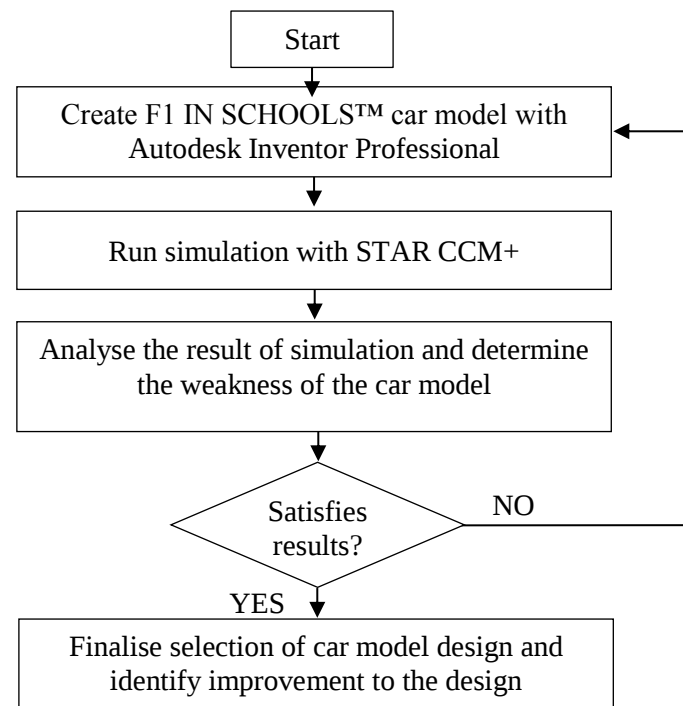
wing design. According to the result of an experiment (Zerihan & Zhang, 2000; Zhang & Zerihan, 2003), a dual element wing is able to create higher downforce compared to the single element wing. However, it also causes higher drag force as shown in Figure 4.

Dual element wing is able to create a higher downforce because it allows the air with higher pressure at the upper surface of the bottom wing to flow into the bottom surface of the upper wing. Thus, the velocity of airflow at the bottom surface of the upper wing is increased. This will result in lower pressure at that particular region (Damjanović et al., 2011). Therefore, the pressure difference between the upper surface and the lower surface of the dual element wing creates a significant downforce on the car body. Overall, inverted wing and dual element wings will be used in this study and will serve as the fundamental design for the front wing. This is because the inverted wing can create a downforce that is essential for the F1 IN SCHOOL™ car. Dual element wing is used as it is able to create a higher downforce. The only drawback of a dual element wing is that it also creates a high drag force on the car body.

Thus, this study will be focusing on the design of the front wing so that it can satisfy two criteria – to produce an F1 IN SCHOOL™ car with sufficient downforce coefficient and low drag coefficient in order to enhance the performance of the F1 IN SCHOOL™ car in terms of speed and stability. Moreover, necessary modification on the car body will be carried out in order to address the limitation encountered by the dual element wing when it is used as a front wing of F1 IN SCHOOL™ car. Furthermore, this study will use the STAR CCM+ to carry out the simulation on the F1 IN SCHOOL™ car. The simulation will then be used to obtain and compare the drag and downforce coefficient.

## 2.0 METHODOLOGY

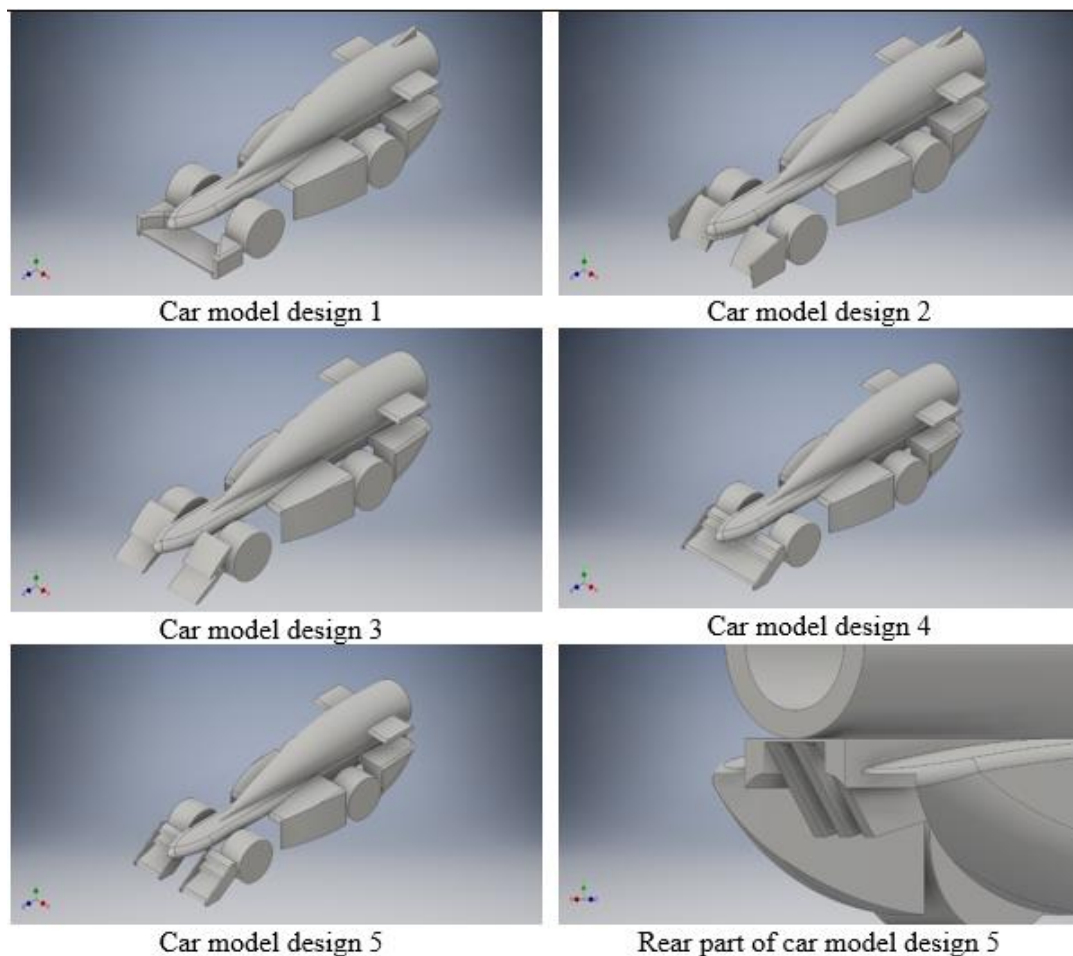
Method that is used as a guide for this study is shown in the flowchart presented in Figure 5.



**Figure 5:** Methodology flowchart

## 2.1. F1 IN SCHOOLS™ front wing design

In this study, the design of the car body is provided by the F1 IN SCHOOLS™ team of PermataPINTAR™ UKM. The focus of this study is to design the front wing that will then be simulated in STAR CCM+. The design process begins with designing the first wing model and simulating it along with the car body using STAR CCM+. After the results on drag coefficient and downforce coefficient are obtained, the strength and weakness of the design will first be revealed and discussed before proceeding to the second design. The next design will focus on solving the weakness of the previous design and take into consideration the maintenance of the strength of the previous design. The Autodesk Inventor Professional software is used as a design tool in this study due to its user-friendly interface. Figure 6 shows 5 designs of the F1 IN SCHOOLS™ car model used in this study.

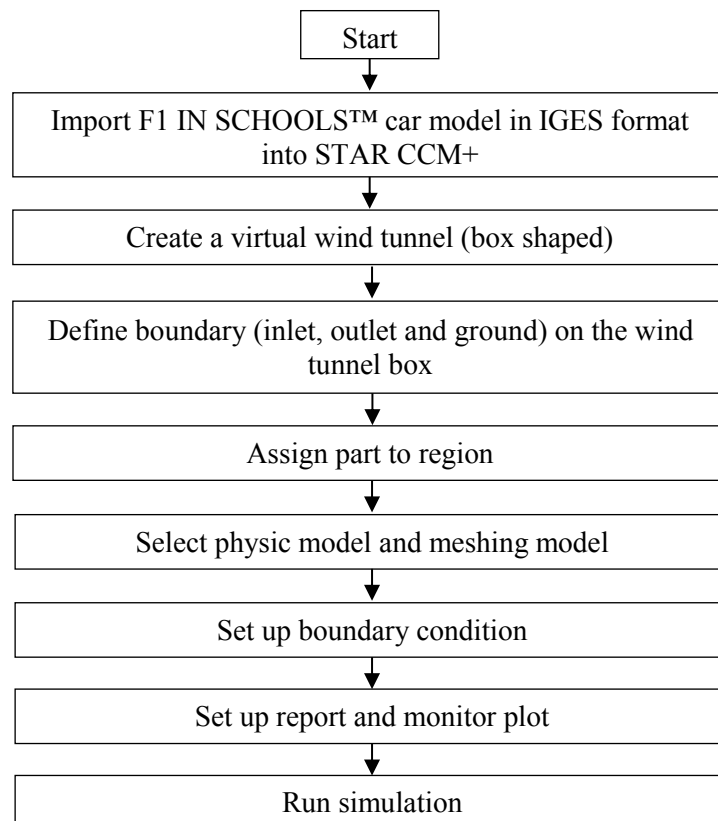


**Figure 6:** Five designs of car model

As seen on the car model designs 1 to 4, the changes were only done at the design of the front wing. Thus, from design 1 to design 4, this study changed the design of the front wing in the spoiler at 1<sup>st</sup> design, to single element wing at 2<sup>nd</sup> design, to dual element wing at 3<sup>rd</sup> design, and to the full span dual element wing with wing support structure at the 4<sup>th</sup> design. Lastly, for the 5<sup>th</sup> design, on top of reducing the span of the wing, an additional modification is made at the underbody of the car model. The underbody is sliced at an angle of 5° and installed with a 2mm thick diffuser, as shown in Figure 6.

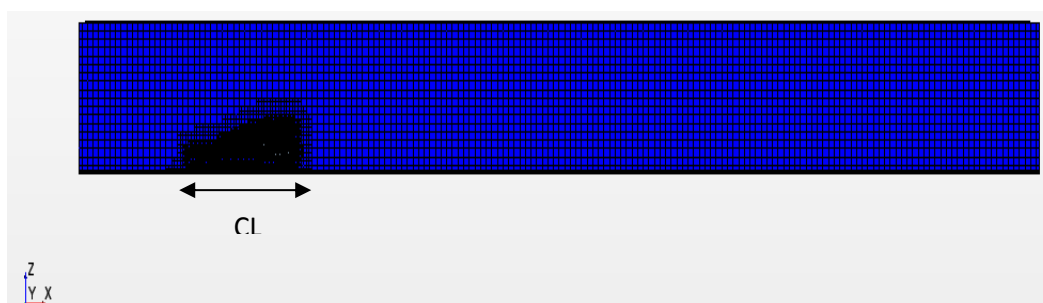
## 2.2. STAR CCM+ simulation

In this study, STAR CCM+ is chosen as the simulation software due to its user-friendly interface. The aim of the simulation is to study the behaviour of external airflow around the car model. This is achieved by observing the velocity streamline of the airflow. Doing so will help determine how the airflow behaviour gives rise to the drag force and downforce of the F1 IN SCHOOLS™ car. Details such as physic model and mesh model in the STAR CCM+ simulation will be discussed in this section. Selection of the physic model and mesh model will eventually affect the result of the simulation due to the different computation methods used. For the STAR CCM+, steps taken to set up the simulation case are shown in Figure 7.



**Figure 7:** STAR CCM+ simulation flowchart

The dimension of the virtual wind tunnel box is "1.71m×0.15m×0.18m" in length, width and height, respectively. The car model is placed at 1 car-length (CL) from the inlet and 7 car-lengths from the pressure outlet. This is shown in Figure 8.

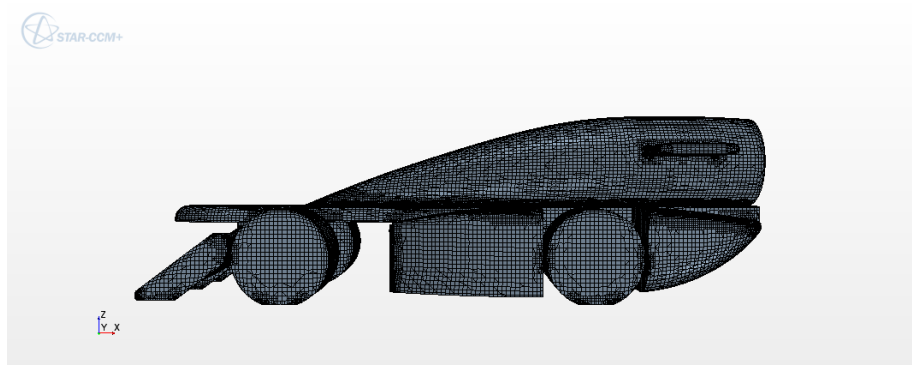


**Figure 8:** Meshed diagram of car and wind tunnel box

In the simulation, the car model is only half of the whole model and it is attached together with the virtual wind tunnel box. This method reduces the complexity of the simulation since meshing of the whole model would need a large memory. Additionally, the symmetry plane was chosen to show the result on another part of the body since they are symmetrical. Next, at the stage of selecting physic and meshing models, model selection is vital since it is related to the method of computation. The reasons for selecting each model in this study are shown in Table 1 below.

**Table 1:** Reason of selection of physic model and meshing model

| Continuum     | Type                         | Description                                                                                                                                                                                                                                                                                                        |
|---------------|------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Physic model  | K- $\omega$ turbulence model | <ul style="list-style-type: none"> <li>• Similar to K-<math>\epsilon</math> turbulence model</li> <li>• Increases the performance of boundary layer that is affected by adverse pressure gradient</li> <li>• Suitable to use for all boundary layer included at boundary that is dominated by viscosity</li> </ul> |
|               | Segregated flow              | <ul style="list-style-type: none"> <li>• Requires low memory</li> <li>• Suitable to use for incompressible flow</li> <li>• Suitable to use at flow with constant density</li> </ul>                                                                                                                                |
|               | Cell quality remediation     | <ul style="list-style-type: none"> <li>• Contributes to flow solution of low quality cell</li> <li>• Applicable at low quality cell only in order to prevent inaccuracy of flow solution</li> </ul>                                                                                                                |
| Meshing model | Surface remesher             | <ul style="list-style-type: none"> <li>• Increases the quality of the surface</li> <li>• Optimum surface for volume meshing</li> </ul>                                                                                                                                                                             |
|               | Trimmer                      | <ul style="list-style-type: none"> <li>• Able to produce high quality grid during meshing simple and complex model</li> <li>• Suitable for use for modelling of aerodynamic external flow because it is able to refine cells in a wake region</li> </ul>                                                           |
|               | Prism layer mesher           | <ul style="list-style-type: none"> <li>• Increases the accuracy of flow solution</li> <li>• Enables flow solver to accurately solve near-wall flow</li> <li>• Able to determine flow separation</li> </ul>                                                                                                         |



**Figure 9:** Car model after meshing

After the physic model and meshing model are selected, the surface mesh and volume mesh are generated. The diagram shown in Figure 9 is the car model used in the simulation after the meshing process. It should be noted that the meshing of the model should be done after the surface repair tool is executed. This is done to ensure that our geometry does not have any feature errors that might lead to any error in the meshing process. Then, boundary condition and air properties have to be defined before the simulation can be carried out. Air properties such as density and viscosity must be defined along with the material properties so that they can be used to compute for drag and downforce coefficient. Moreover, the boundary conditions at inlet, outlet and car body must be defined before proceeding to the simulation. All the boundary conditions and their respective values have been tabulated and shown in Table 2 below.

**Table 2:** Boundary condition at STAR CCM+

| Boundary/Fluid       | Parameter                    | Values                 |
|----------------------|------------------------------|------------------------|
| Air properties       | Density (kg/m <sup>3</sup> ) | 1.177                  |
|                      | Dynamic viscosity (Pa-s)     | $1.846 \times 10^{-5}$ |
|                      | Temperature (K)              | 300                    |
| Velocity inlet       | Velocity (m/s)               | 22.2222                |
|                      | Gauge pressure (Pa)          | 0                      |
|                      | Turbulence viscosity ratio   | 100.0                  |
|                      | Turbulence intensity         | 0.03                   |
| Pressure outlet      | Gauge pressure (Pa)          | 0                      |
|                      | Turbulence viscosity ratio   | 100.0                  |
|                      | Turbulence intensity         | 0.03                   |
| Ground               | Shear stress specification   | Slip condition         |
| Front and rear wheel | Angular velocity (rad/s)     | 793.65                 |

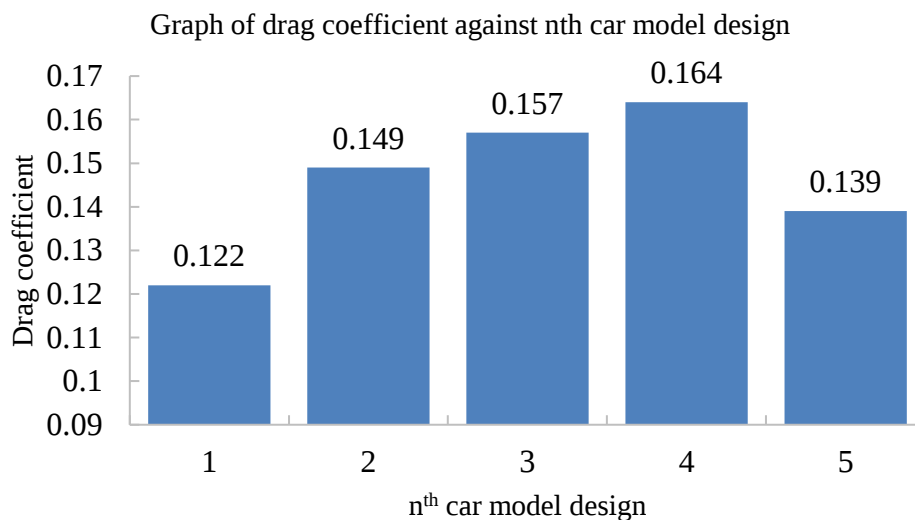
The air properties were recorded based on the assumption that it is at room temperature and at an absolute pressure of 1atm. Note that the pressure column in STAR CCM+ is based on the gauge pressure instead of the absolute pressure. The ground region is set as slip condition so that the software is able to simulate it to the real case where the car is seen as moving on the ground. Moreover, in the setting of STAR CCM+, the wheel of the car model can be simulated as if it is in rotation at a specific angular velocity. This ensures that the result obtained from the simulation does not have a lot of difference compared to its real life application. These properties and boundary conditions remain the same for all 5 designs of the car model in the STAR CCM+ simulation. This is due to the fact that this study aims to examine the effect of the front wing design on the drag and downforce coefficient, but not the effect of the properties on the parameter.

After all the settings have been completed, the plot on drag coefficient and lift coefficient needs to be created so that the simulation can compute and record the result of the drag and lift coefficient and use it for further analysis. Note that in this study, the direction of the drag is set at positive x-direction and direction of the lift is set at positive z-direction. However, its direction should depend on the direction of the car's placement in STAR CCM+.

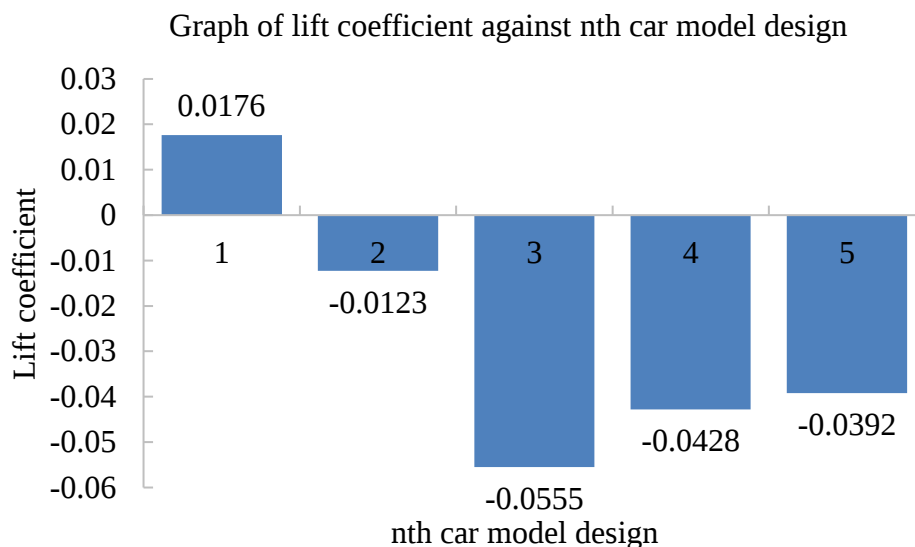
### 3.0 RESULTS AND DISCUSSION

From the simulation, the results of drag coefficient and lift coefficient for all car model designs have been calculated. Graphs of drag coefficient and downforce coefficient against the nth

number of car model design are plotted and shown in Figure 10 and Figure 11, respectively. The graph in Figure 11 shows that the drag coefficient increased from 0.122 at the car model design 1 to 0.164 at the car model design 4. This is an increment of 34.4%. However, it successfully reduced to 0.139 for the car model design 5, which corresponds to a decrement of 15.2%. Based on the result, two main factors that contribute to an increase in drag coefficient are the frontal area of the front wing and the flow separation zone at the rear part of the car model. As shown in Figure 6, the frontal area of the car model increases as different front wing designs are attached to the car body. As the frontal area increases, more airflow will be resisted by the front wing. This will result in an increased drag force that will be applied on the front wing. According to Equation 1, as the drag force increases, the drag coefficient will also increase.

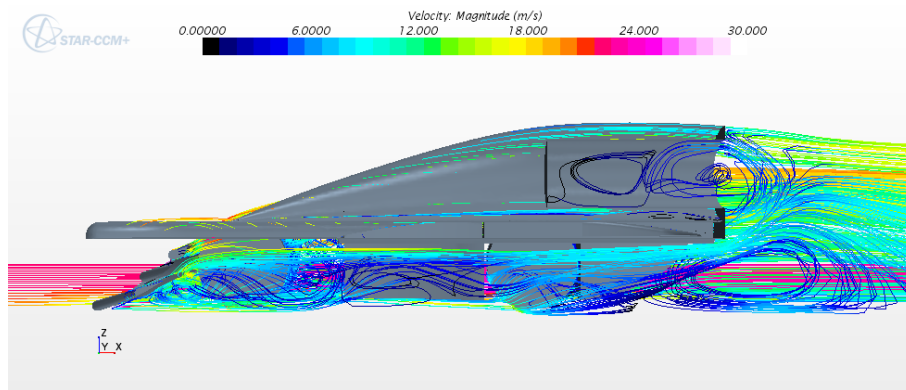


**Figure 10:** Graph of drag coefficient against nth car model design

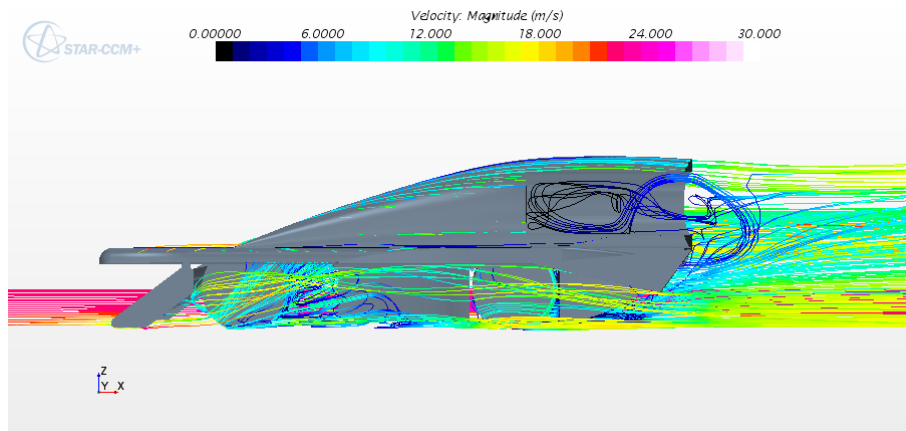


**Figure 11:** Graph of lift coefficient against nth car model design

Besides, the size of flow separation zone will also affect the drag coefficient. Its effect on drag coefficient is more significant for the car model design 4, as shown in Figure 12. The airflow experiences a large flow separation zone at the rear part of the car. Therefore, a low pressure zone is created at that particular region. The increase in pressure difference between the front and rear part of the car will result in a higher drag force. The decrement in drag coefficient recorded from car model design 4 to car model design 5 is a result of the modification done on the car body. For car model design 5, the car body is slicing at  $5^\circ$  horizontally and a diffuser measuring 2mm is added to the underbody. This feature has successfully directed the airflow at the underbody of the car to the low pressure zone at rear part. Therefore, as shown in Figure 13, the size of flow separation zone is reduced.



**Figure 12:** Velocity streamline at underbody and rear part of car model design 4

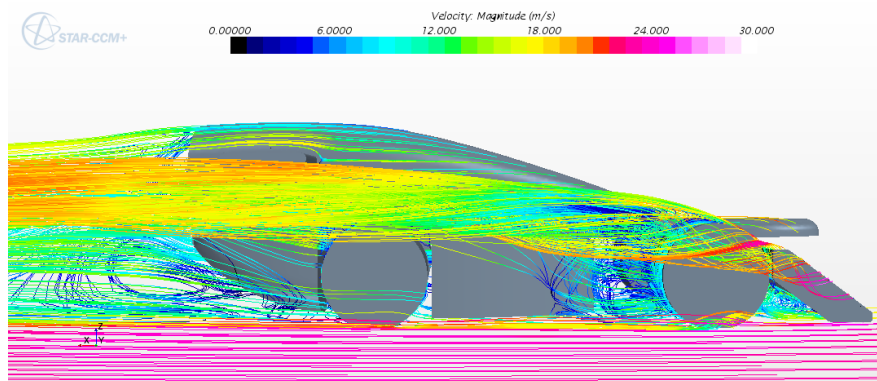


**Figure 13:** Velocity streamline at underbody and rear part of car model design 5

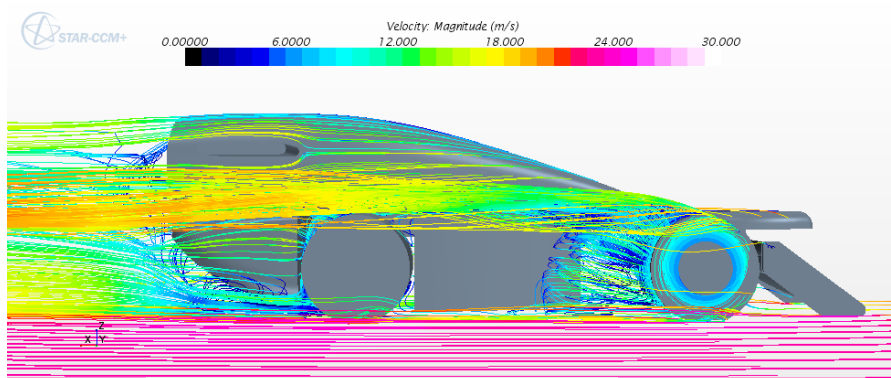
Since the size of the separation zone has decreased, it will reduce the pressure difference between the front and rear part of the car. Thus, the drag force applied on the car body will decrease. Eventually, the drag force decreases. From Figure 13, it can be seen that the lift coefficient changes from positive to negative from car model design 1 to car model design 2. This indicates that the normal force acting on car model design 1 is a lift force, whereas the normal force acting on car model design 2 is a downforce. Moreover, the downforce coefficient increases from 0.0123 to 0.0555 for car model designs 2 and 3, respectively. Then, it decreases to 0.0392 for car model design 5. The factor that contributes to the increase is the changing of the lift force to downforce from car model design 1 to car model design 2. This is a result of the installation of a single element wing. A single element wing will generate downforce along

with the aerodynamic ground effect. When a single element wing is attached to the car body, the ground clearance is reduced. This results into airflow acceleration at the underbody of the car. Therefore, the difference in pressure between the upper and bottom parts has generated downforce on the car. The continued increment of downforce coefficient from car model design 2 to car model design 3 is due to the replacement of the single element wing with a dual element wing. This is due to the fact that a dual element wing can produce a greater pressure difference between the upper airflow and bottom airflow. In turn, this contributes to the increment in downforce coefficient. Although the downforce coefficient experiences a slight decrement from car model design 3 to car model design 5, it is still enough to ensure that the car's wheel is always in contact with the track surface.

The decrement in downforce coefficient from car model design 3 to car model design 4 is mainly due to the fact that the ground clearance at car body design 4 is lower than the optimum ground clearance. However, the decrement of downforce coefficient from car model design 4 to car model design 5 is mainly due to the airflow on top of the car surface. This airflow is directed by the front wing. Figure 14 and Figure 15 clearly show that the airflow on top of car model design 4 is more than the airflow at the upper surface of car model design 5. This implies that the air pressure at the upper surface of car model design 5 is lower than the airflow of car model design 4. Therefore, the downforce of car model design 5 is reduced. Although the downforce on car model design 5 was reduced, it is still enough to ensure that the car's wheel is always in contact with the road surface.



**Figure 14:** Velocity streamline at the upper surface of car model design 4



**Figure 15:** Velocity streamline at the upper surface of car model design 5

#### 4.0 CONCLUSIONS

Based on the result of drag coefficient and downforce coefficient discussed in the previous section, the car model design 5 is selected as the F1 IN SCHOOLS™ car since it is able to generate a low drag coefficient of 0.139 and a downforce coefficient of 0.0392. Based on Equation 1 and Equation 2, the drag force and downforce applied on the car body is approximately equal to 0.1926N and 0.05431N, respectively.

In this study, the two main focuses of aerodynamic study on F1 IN SCHOOLS™ car are drag coefficient and downforce coefficient. Downforce on a car model can be created by using the inverted wing as front wing. Both the single element wing and dual element wing can be used since they have the same behaviour in creating downforce. However, the dual element wing has better performance since it creates a greater downforce compared to the single element wing. As stated by Katz (2006), although downforce is beneficial to the stability of F1 In SCHOOLS™ car, it also gives rise to drag force. To reduce drag force, modifications on the car body can be carried out. Examples of modification include slicing the rear underbody of the car at a particular horizontal angle and installation of the diffuser. Both these efforts make it possible to direct the airflow from the underbody of the car to the flow separation zone. Thus, the pressure difference between the front and rear part of the car model is reduced.

Lastly, this study can be further improved through two approaches. In this study, focus is given on the performance of front wing in terms of drag and downforce coefficient. The aim is to increase the performance of F1 IN SCHOOLS™ car based on speed and stability. Other than focusing on the front wing, the first way to improve this study is to identify and reduce the friction between the car's wheel and the surface of the track since it is one of the factors that contributes to drag coefficient. The second approach that can be applied to improve this study is in terms of launching energy recovery system (LERS) of F1 IN SCHOOLS™ car. The current LERS technology has a limitation because carbon dioxide gas has a tendency to diverge at the point of launching. This results into low kinetic energy transfers to the car and thus reduces its speed during launching. Therefore, efforts can be focused on investigating and creating a new mechanism for the LERS system on F1 IN SCHOOLS™ car.

#### ACKNOWLEDGEMENTS

This research was supported by STEM-2014-012 grant.

#### REFERENCES

- Damjanović, D., Kozak, D., Živić, M., Ivandić, Ž., & Baškarić, T. (2011). CFD analysis of concept car in order to improve aerodynamics. *A JOVO jarmuve, A MAGYAR JÁRMŰIPAR TUDOMÁNYOS LAPJA*, 1(2), 63-70.
- Guilmineau, E. (2008). Computational study of flow around a simplified car body. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(6–7), 1207–1217.  
doi:10.1016/j.jweia.2007.06.041

- Hetawal, S., Gophane, M., Ajay, B. K. & Mukkamala, Y. (2014). Aerodynamic study of formula SAE car. *Procedia Engineering*, 97, 1198–1207. doi:10.1016/j.proeng.2014.12.398
- Katz, J. (2006). Aerodynamics of race cars. *Annual Review of Fluid Mechanics*, 38(1), 27–63.
- Kuya, Y., Takeda, K., Zhang, X., Beeton, S. & Pandaleon, T. (2009). Flow separation control on a race car wing with vortex generators in ground effect. *Journal of Fluids Engineering, Transactions of the ASME*, 131(12), 1211031–1211039. doi:10.1115/1.4000420
- McCabe, G. (2008). *Explanation and discovery in aerodynamics*. Retrieved from <http://arxiv.org/abs/physics/0512224>
- Rakibul Hassan, S. M., Islam, T., Ali, M. & Islam, M. Q. (2014). Numerical study on aerodynamic drag reduction of racing cars. *Procedia Engineering*, 90, 308–313. doi:10.1016/j.proeng.2014.11.854
- Reddy, J. J. & Mech, B. T. (2006). Finding the optimum angle of attack for the front wing of an F1 car using CFD. *Proceedings of the 4<sup>th</sup> WSEAS International Conference on Fluid Mechanics and Aerodynamics*, 29–34.
- Rozhdestvensky, K. V. (2006). Wing-in-ground effect vehicles. *Progress in Aerospace Sciences*, 42(3), 211–283.
- Soso, M. D. & Wilson, P. a. (2006). Aerodynamics of a wing in ground effect in generic racing car wake flows. *Journal of Automobile Engineering*, 220, 1–13. doi:10.1243/095440705X69632
- Vogt, J. W. & Barber, T. J. (2012). Ground effect phenomena about lift and downforce generating cambered aerofoils. *International Journal of Numerical Methods for Heat & Fluid Flow*, 22(2), 153–174.
- Zerihan, J. & Zhang, X. (2000). Aerodynamics of a single element wing in ground effect. *Journal of Aircraft*, 37(6), 1058–1064.
- Zhang, X., Toet, W. & Zerihan, J. (2006). Ground effect aerodynamics of racing cars. *Applied Mechanics Reviews*, 59(1–6), 33–48. doi:10.1115/1.2110263
- Zhang, X. & Zerihan, J. (2003). Aerodynamics of a double element wing in ground effect. *AIAA Journal*, 41(6), 1007–1016.