Volume 4 Issue 1 (March 2024)
DOI 10.24191/jaeds.v4i1.77
eISSN 2805-5756

Journal of Applied Engineering Design & Simulation

Optimising F1 in Schools Car Design: Using CFD Simulation and Wind Tunnel Testing for Enhanced Aerodynamic Performance

M.H. Muhammad[1]*, M. N. A. M. Den[1], R. Kasiran[1] , N. Ikhsan[1]
[1]School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, Shah Alam 40450 Selangor, Malaysia.
*corresponding author: mhanif76@uitm.edu.my

ABSTRACT

F1 in Schools, a STEM competition for students, fosters collaborative design and manufacturing of the fastest miniature cars inspired by Formula 1 racing. To achieve the fastest car, the design must be optimised to reduce drag as much as possible. This paper focuses on optimising drag force, a key determinant of speed and stability. Through CFD simulations, various design iterations were evaluated. Each change was made based on targeting airflow obstructions and flow separation. A wind tunnel experiment was conducted to verify the results obtained through CFD. Results show that modifications significantly reduced drag force by 9.89%. Insights from this study underscore the importance of iterative design processes. Further enhancements could involve analysing pressure distribution and lift force to maximise thrust utilisation and improve race performance.

Keywords: CFD; F1 in Schools, drag force, aerodynamic, wind tunnel.

Nomenclature (Greek symbols towards the end)

L	car length
H	car height
W	weight of the car
F_T	thrust force
F_D	drag force
F_{RR}	rear rolling resistance
F_{RF}	front rolling resistance
F_L	total lift force
a	acceleration
m	mass
C_D	coefficient of drag
V	velocity
ρ	air density

Abbreviations

STEM	science, technology, engineering, and mathematic
3D	three dimensional
CAD	computer-aided design
CAM	computer-aided manufacturing
CFD	computational fluid dynamics
CG	centre of gravity
CP	centre of pressure

1.0 INTRODUCTION

F1 in Schools is an international STEM competition for students aged 11 to 19, where teams of 3 to 6 collaborate to design and manufacture miniature cars inspired by Formula 1 racing. Key aspects include using three-dimensional (3D) computer-aided design (CAD) software for design, analysing aerodynamics through computational fluid dynamics (CFD), employing 3D computer-aided manufacturing (CAM) software for manufacturing, conducting wind tunnel tests, and competing to achieve the fastest and most efficient design.

The size of the F1 in Schools is around 1:24 on the scale of the actual F1 car and is raced on a 20 m long straight track. It's powered by an 8 g compressed CO2 cartridge that is pierced using a launcher to generate thrust at the rear of the vehicle. The race method in F1 in Schools involves teams competing against each other to see who can achieve the fastest time down the track. There are many significant engineering forces involved in the

Received on 19.03.2024
Accepted on 27.03.2024
Published on 30.03.2024

F1 in schools cars, as shown in Figure 1. Forces including drag, lift, weight, rolling resistance and thrust all play a role in determining the speed and stability of the car.

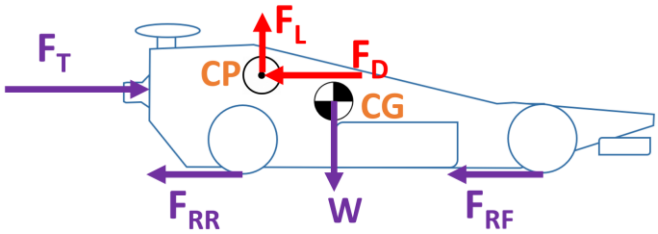


Figure 1. Forces acting on F1 in Schools car during the race

The aerodynamics of the F1 in Schools car play a crucial role in its performance. The design of the car's body, wings, and other components are carefully engineered to minimise air resistance and optimise lift force. To achieve this, teams utilise computational fluid dynamics software to analyse airflow around the car and make design modifications accordingly. Applying the second law of Newton to the horizontal force in Fig. 1 gives:

$$\sum Forces = m.a \qquad \text{(N)} \qquad (1)$$

$$F_T - F_D - F_{RR} - F_{RF} = m.a \qquad \text{(N)} \qquad (2)$$

By rearranging the equation (2), the acceleration of the car is:

$$a = \frac{F_T - (F_D + F_{RR} + F_{RF})}{m} \qquad (m/s^2) \qquad (3)$$

From Equation 3, it can be observed that the acceleration of the car is dependent on the drag force, as well as the rear and front wheel rolling resistance. In this paper, we focus on optimising the drag force since it directly opposes the vehicle. This is because the thrust force is generated from the canister given by the organiser, and the mass can be minimised by creating a design with the minimum weight allowed. As for the front and rear rolling resistance, this involves the design of the wheel and wheel loads, which is not in the scope of analysis [1].

The drag force is dependent on several factors, mainly the drag coefficient (C_D) and the frontal area (A) like, as shown in Equation 4 below:

$$F_D = \frac{1}{2}\rho C_D A V^2 \qquad \text{(N)} \qquad (4)$$

Since every car will be racing in the same location, the air density will be constant for all participants. Here, the velocity (V) of the analysis will be based on the fastest record time achieved at the world finals, 0.916 s [2]. Since the analysis of the model was made before the introduction of Halo, the model in this paper does not take this into account [3]. Findings show older F1 in School models can achieve a drag coefficient below 0.15 [4]. Still, newer regulations are added from one year to another to challenge school students to be more creative in solving aerodynamic problems.

2.0 METHODOLOGY

2.1 Computational Fluid Dynamics

Using CFD will simplify the study of determining the drag force of the complex shape of F1 in Schools car. Figure 2 shows the methodology of the CFD process. For this particular study, Ansys Student 2023 R1 is used, and it is limited to 512K cells/nodes.

The fluid domain for this particular car will be positioned within a virtual wind tunnel measuring 2H x 2W x 9L, proportionate to the model's dimensions, to ensure that the airflow fully develops, as shown in Fig. 3 [5]. As for the turbulent model, $k - \omega$ was selected to reduce computational time and is suitable for a slender body like the F1 in Schools car [6, 7]. The fluid used is air at room temperature, set to begin at the inlet at 22 m/s; this value is based on the average speed set by the world record discussed earlier. For the remaining boundary, the outlet is set to a pressure gauge of 0 Pa and the wall is set to slip to simulate a moving car. Here, due to the student version limitations, the virtual wind tunnel is cut in half to reduce the number of cells. Since the vehicle is symmetric, this doesn't affect the result for steady state analysis, but the symmetry boundary condition must be set on the symmetry wall [8]. Figure 4 shows a diagram of the boundary applied in the model. The overall car dimension is 196 mm in length, 61 mm in height and 68 mm in width, with a ground clearance of 5 mm.

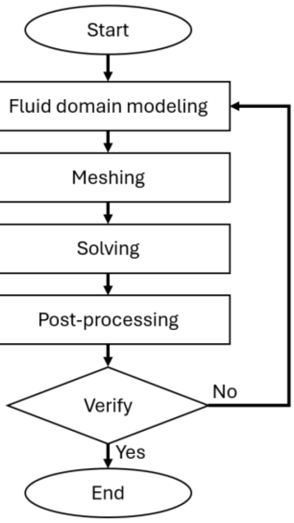
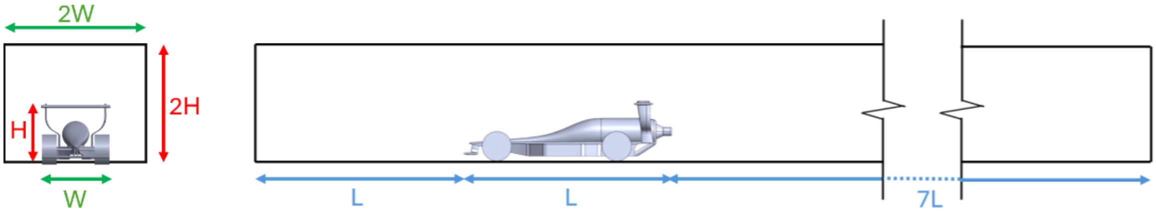


Figure 2. CFD simulation methodology



(a) Front view (b) Side View

Figure 3. Virtual wind tunnel size

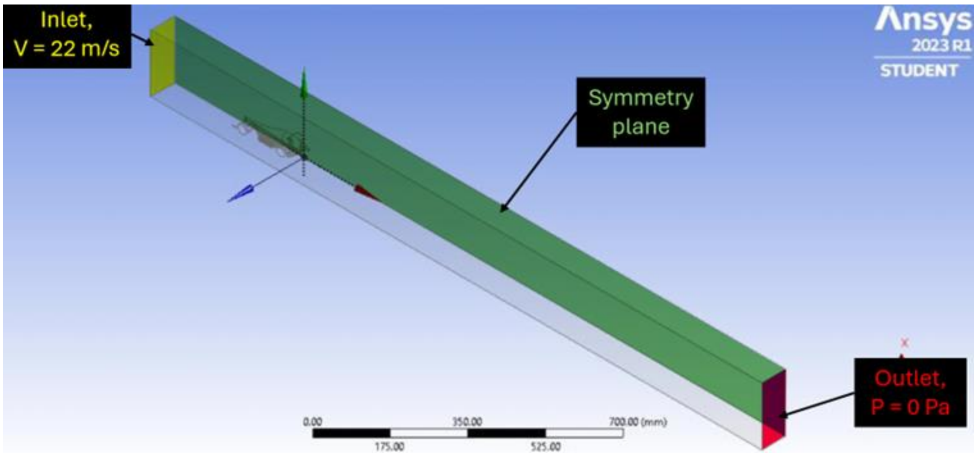


Figure 4. Boundary conditions

The boundary was mesh using the default setting of unstructured tetrahedral mesh. Due to the limitation of the student model, further refinement cannot be done. For the same reason, grid dependant study was not executed. The mesh generated around vehicle is shown in Fig. 5. The CFD process was repeated several times to optimise the design of the car. Optimisation involved identifying problem areas and redesigning the car to decrease or eliminate aerodynamic issues. The differences between the three simulated car models can be seen in Fig. 6. The reason behind the changes is explained in the result section. Model 2 differs from model 1 by hollowing the side pod. The last model was modified by increasing the ground clearance at the centre of the body.

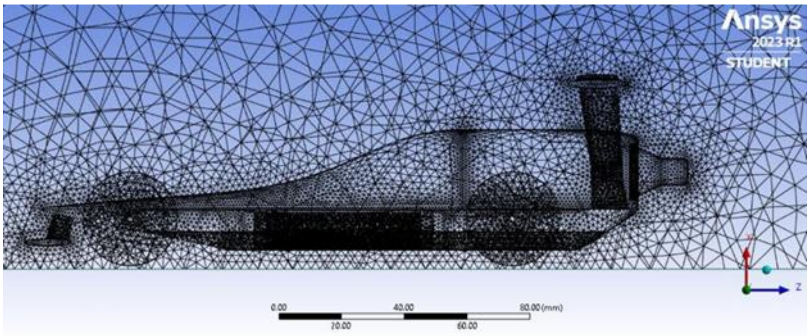
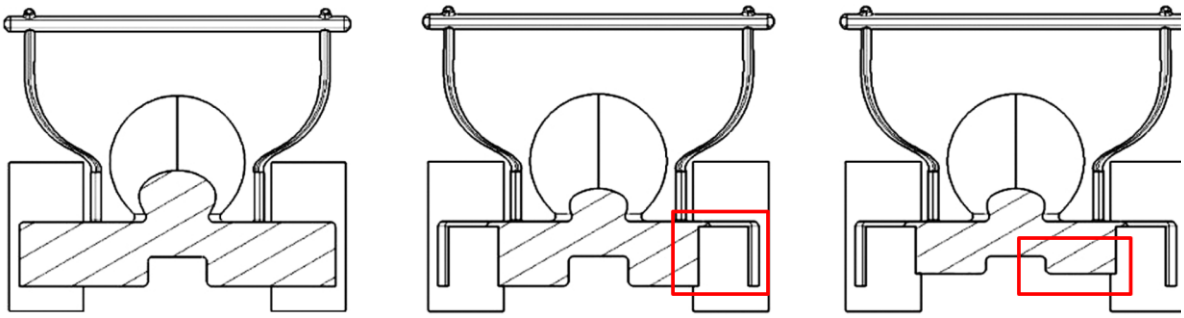


Figure 5. Mesh around F1 in the Schools car



(a) Model 1 (b) Model 2 (c) Model 3

Figure 6. The cross-section of the model shows the change in the design of models 1, 2, and 3.

2.2 Wind tunnel experiment

A wind tunnel experiment was made to verify the findings made through CFD. However, the setup is different since the wind tunnel use does not put the car on the ground like in the CFD domain. The car position and the mounting rod are shown in Fig. 7. The low-speed wind tunnel model is 400mm x 300mm x 300 mm, big enough to have a fully developed flow. The speed was set to 22 m/s, the same as the CFD simulation, and the car was positioned in the middle of a wind tunnel. Only the final design was tested for verification. Since the vehicle is mounted on the rod, the drag force on the rod must be taken out of the total drag force. This is done by running the wind tunnel only with the rod present.

Figure 7. Position of F1 in school car model at the centre of Wind tunnel testing section

3.0 RESULTS AND DISCUSSION

3.1 Computational Fluid Dynamics

Figure 8 shows the velocity profile of the overall simulation. The objective here was to verify that the virtual wind tunnel was big enough to have a fully developed region. As shown in the figure, at the rear and the top of the vehicle, there is enough space for the air wake to recuperate. This is important so that our result is not affected by the friction of the boundary walls.

For the first iteration, the final concept car model 1 was used, as shown in Fig. 9 (a). Figure 10 shows the side view velocity plot at the middle plane of the wheels. The resistance created by the obstruction of the sidepod causes the air to flow around it. The flow in front and rear of the sidepod has flow separation generating a low presurre votices. The same phenomena can also be observed at the rear-end diffuser of the car. The velocity plot from the top view shown in Fig. 10 shows the vortices are more prominent at the rear of the sidepod. Due to these flow patterns, the overall car drag force was 0.2517 N. It is to be noted that due to the open wheel design regulation, the wheel will play a significant problem in optimising the drag force of the car [9].

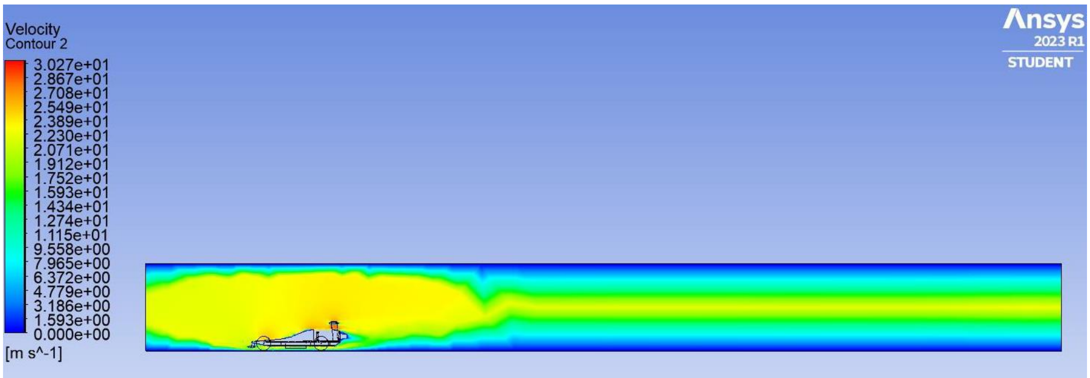
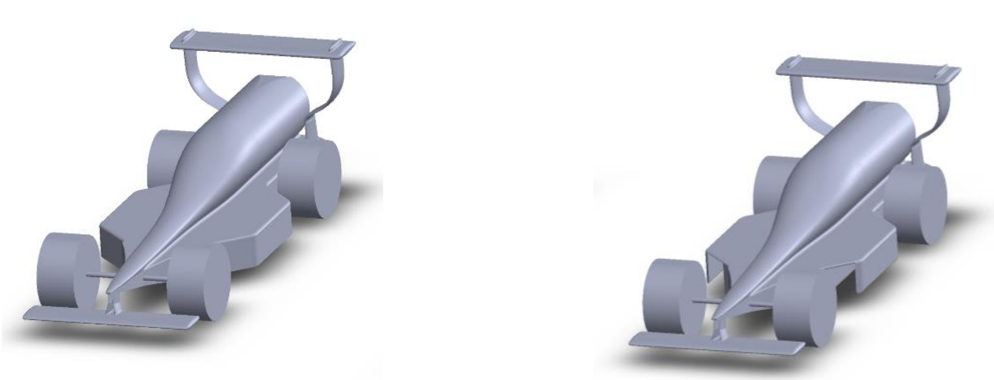


Figure 8. External velocity contour around F1 in Schools car



(a) F1 in Schools car model 1 (b) F1 in Schools car model 2

Figure 9. F1 in Schools car models used in the simulation

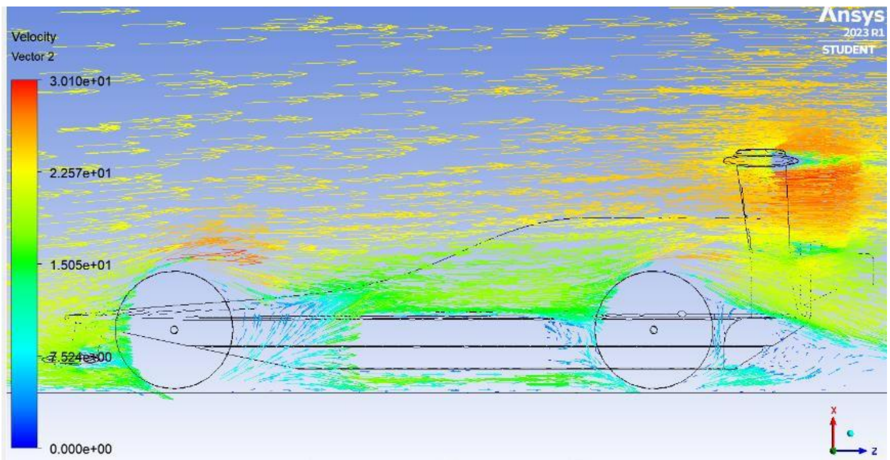


Figure 10. F1 in Schools car models 1 side view velocity plot

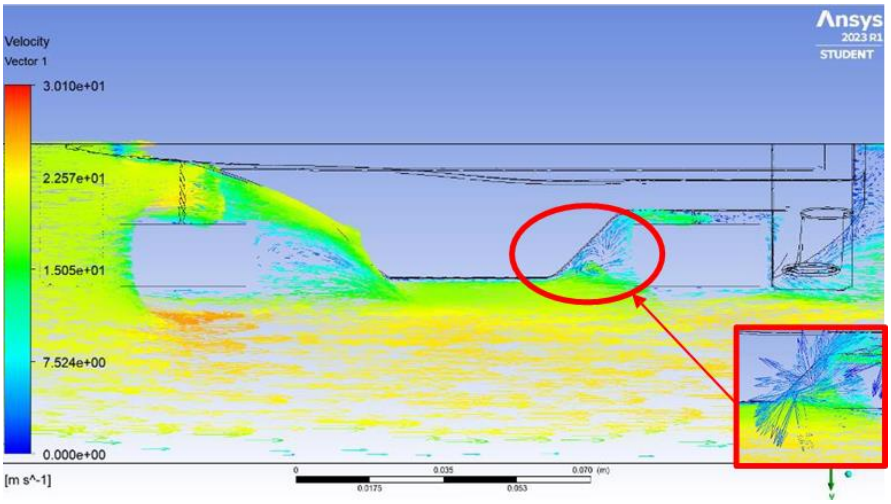


Figure 11. F1 in Schools car model 1 top view velocity plot

In order to reduce the obstruction of the sidepod, the second design shown in Fig. 9 (b) has the sidepod hollowed on the bottom side. From the result shown in Figs. 12 and 13, the flow separation that was due to the sidepod obstruction is significantly reduced. The velocity contrast between the inlet and outlet of the sidepod remains nearly constant, promoting a straight-flowing direction that ensures smooth airflow across the entire sidepod region. However, the smooth flow around the sidepod has increased the vortices around the rear diffuser. Nevertheless, the overall drag force has been reduced to 0.2456 N. This made Model 2 have a 2.4% drag force decrease compared to Model 1. The hollow section of the sidepod does not affect the flow of the other components of the car.

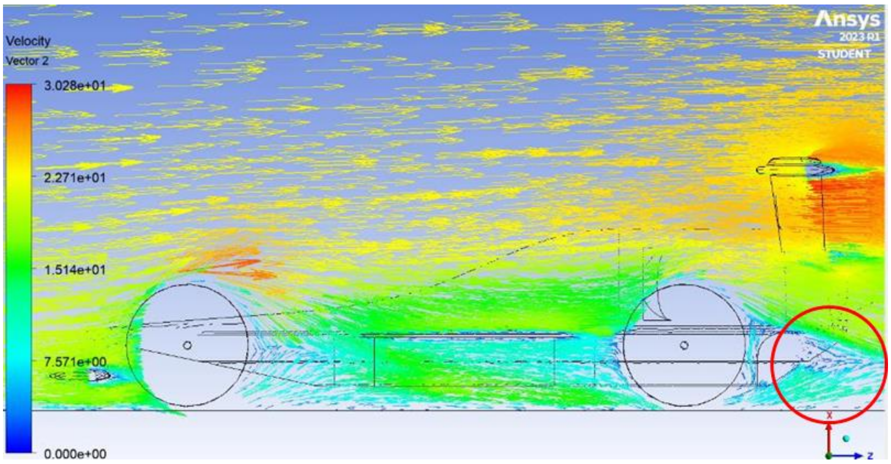


Figure 12. F1 in Schools car model 2 side view velocity plot

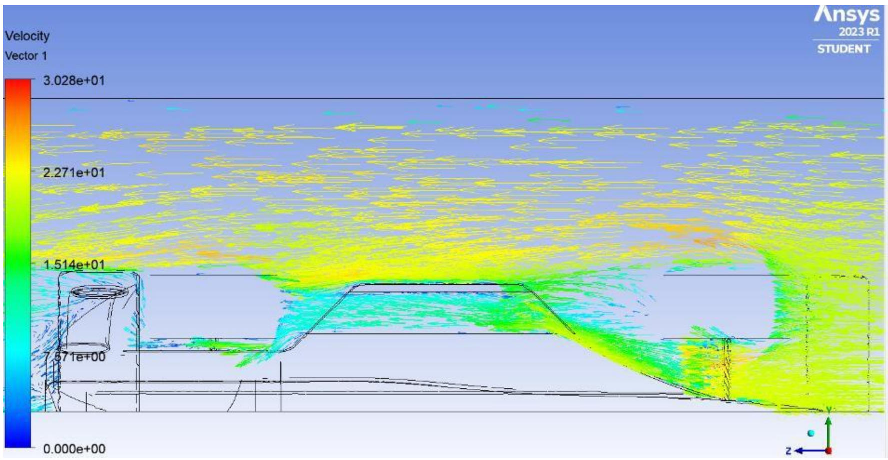


Figure 13. F1 in Schools car model 2 top view velocity plot

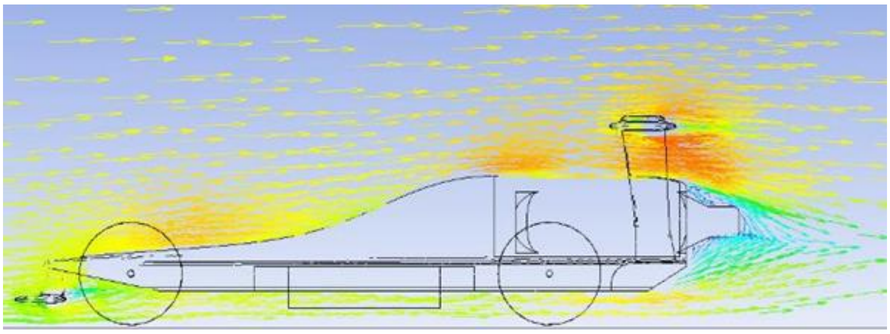
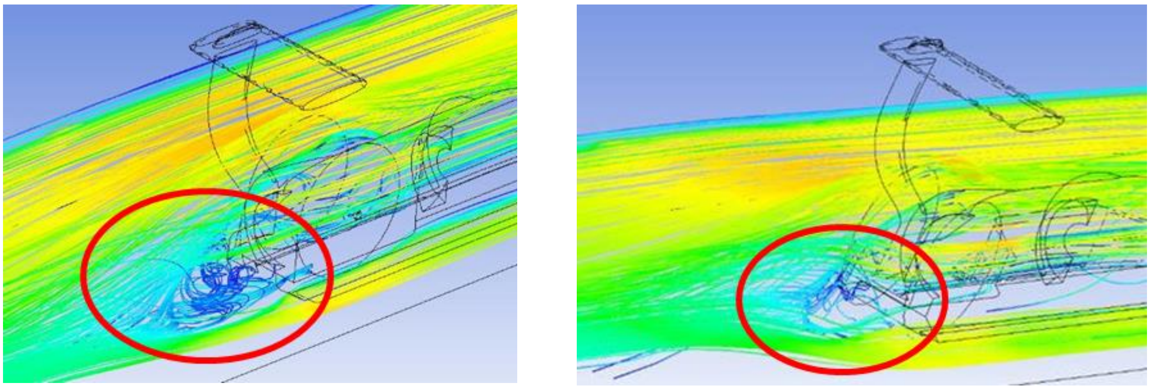


Figure 14. F1 in Schools car model 3 side view velocity plot

For the final improvement, the focus was to reduce the problematic flow at the rear diffuser. This was done by increasing the ground clearance by 5 mm. The result shown in Fig. 14 is that the modification of the rear diffuser helps to reduce flow separation between the front and rear parts of F1 in the school car model. This helps to direct airflow under the car body directly into the low-pressure zone at the rear part of the car model. Referencing Fig. 15, the flow separation for model 3 is less intense and smaller compared to the flow separation for models 1 and 2. This reduction of flow separation size helps to reduce the drag coefficient. The modification resulted in a decrease of drag force by 7.56% at 0.2268 N.



(a) Model 2 (b) Model 3

Figure 15. Vortex at the rear diffuser

Table 1: CFD result summary

Model	Drag Force (N)	Improvement from the previous model
1	0.2517	-
2	0.2456	2.42 %
3	0.2268	7.65 %

Table 2: Wind tunnel result summary

Exp. no	Drag force, [Car + Rod] (N)	Drag force, [Rod] (N)	Drag force, [Car] (N)
1	0.460	0.157	0.303
2	0.466	0.150	0.316
3	0.470	0.154	0.316
	Average of drag force of F1 in school car (N)		**0.312**

3.2 Wind tunnel experiment

The final model was tested in the wind tunnel, and the summary of the experiment is tabulated in Table 2. Three separate runs were made, and the average drag force averaged at 0.312 N. The difference in value is due to the difference in setup. The car positioned in the middle of the wind tunnel reduces the ground effect compared to the simulation. The difference in value is 27%; conversely, this is consistent with a finding that for each 1% of ground clearance, the drag force will increase by 0.28% [10]. Another finding involving the F1 car wing also finds that increasing ground clearance by 82% increases drag by 30%.

4.0 CONCLUSION

From the CFD simulations, the optimisation was achieved by reducing the total drag force by 9.89% from the original model 1. By visualising the flow result, problematic areas can be determined, and it is critical to make modifications to the car. From the modification on model 2, it can be observed that changes in another area of the car will have an impact on another part of the car. It is prudent to solve one part of the problem one at a time. Setting up the CFD analysis to mimic the experimental wind tunnel would have eased the comparison but would not show the car's performance on the track. From the analysis, the main contributor to the reduction of drag was the increase in ground clearance in Model 3.

The findings in this paper could be improved by looking at the pressure distribution on the car. This can indicate where the design of the vehicle can obstruct the flow path. By looking at the centre of pressure and lift force, the race time can be improved by making sure the car launches properly, maximising the use of thrust force from the canister. Due to the open wheel design, achieving low drag force like the model with older regulation is quite challenging. The addition of Halo for the new rules increases the challenges of optimising the aerodynamics of the F1 in Shools cars.

ACKNOWLEDGEMENT

The authors would like to extend our gratitude to the academic and technical staff at the School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, Selangor, Malaysia for the support in the completion of the work. Images used courtesy of ANSYS, Inc.

REFERENCES

[1] V. Jansson, "A literature study of rolling resistance and its affecting factors," KTH Royal Institute of Technology, 2022.

[2] F1 in Schools. "Australia set new record at F1 in Schools World Finals." https://www.formula1.com/en/latest/headlines/2016/10/australia-set-new-world-record-at-f1-in-schools-world-finals.html (accessed 19 March, 2024).

[3] "How F1 in Schools is Introducing the Next Generation to Motorsport." https://www.mclaren.com/racing/formula-1/2023/how-f1-in-schools-is-introducing-the-next-generation-to-motorsport/ (accessed 19 March, 2024).

[4] M. R. A. Mansor and Z. Harun, "F1 in Schools competition to promote STEM: Aerodynamic investigation of miniature F1 in Schools car," in *2017 7th World Engineering Education Forum (WEEF)*, 2017: IEEE, pp. 708-712.

[5] A. Guerrero and R. Castilla, "Aerodynamic study of the wake effects on a formula 1 car," *Energies*, vol. 13, no. 19, p. 5183, 2020.

[6] U. Ravelli and M. Savini, "Aerodynamic investigation of blunt and slender bodies in ground effect using OpenFOAM," *International Journal of Aerodynamics*, vol. 7, no. 1, pp. 36-60, 2020.

[7] L. Cattafesta, C. Bahr, and J. Mathew, "Fundamentals of wind-tunnel design," *Encyclopedia of aerospace engineering*, pp. 1-10, 2010.

[8] M. Aultman, Z. Wang, R. Auza-Gutierrez, and L. Duan, "Evaluation of CFD methodologies for prediction of flows around simplified and complex automotive models," *Computers & Fluids*, vol. 236, p. 105297, 2022.

[9] E. Josefsson, T. Hobeika, and S. Sebben, "Evaluation of wind tunnel interference on numerical prediction of wheel aerodynamics," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 224, p. 104945, 2022.

[10] A. Cogotti, "A parametric study on the ground effect of a simplified car model," *SAE transactions*, pp. 180-204, 1998.