VOL. 11, NO. 20, OCTOBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL STUDY OF AERODYNAMIC DRAG FORCE ON STUDENT FORMULA CAR

Shukriwani Saad[1] and Mohd Fadzli Hamid[2]
[1]Faculty of Computer, Media and Technology Management, Department of Technology Management, TATI University College, Kijal, Kemaman, Terengganu, Malaysia
[2]School of Mechanical Engineering, University Science Malaysia, Nibong Tebal, Penang, Malaysia
E-Mail: shukriwani@tatiuc.edu.my

ABSTRACT

The aerodynamic design concept is a one of the forecast objective to enhance the capability towards acceleration thus restricted from undesired phenomena occurs such as wake area, turbulence flow and so forth. The virtual wind tunnel was introduced to contribute the alternative ways to accomplish the analysis. Numerous model was designed and verified via Computational Fluid Dynamics constructed by ICEM/CFD FLUENT which is a CFD solver that be employed in this study. This technique in order to reduce the drag force value meanwhile brought the student formula car has a tendency to streamline expense to reduce the resistance force by the motion of air, more excellent handling and stability performance. From the result has found that the Kolej Komuniti Bandar Darul Aman (KKBD) student formula car is situated in the range of aerodynamics value which 0.28, and adequate significance to perform in Eimarace competition. Virtual simulation study has brought a result where inspired student to increase a confident level, especially in the design concept of student formula car thus would suggest the best design in future development. From the simulation study also can observe the area which generated turbulence flow, low pressure, high pressure, coefficient of drag, the coefficient of lift, wake area, pressure distribution on the surface, trailing vortices flow, noise study, skin friction drag and pressure drag. From the simulation study has found that the separated flow occurs whenever Go-Kart moving in above 50km/h, the more blunt the vehicle will produce the higher coefficient of drag.

Keywords: aerodynamics, CFD, design, coefficient of drag, student formula car.

INTRODUCTION

Aerodynamics study is a study of motion of air, especially when it interacts with a moving or non-moving object. The understanding of the air motion will enable the calculation of forces and moment acting on the object. By defining a control volume around the flow field, the equation for the conservation of mass, momentum and energy can be defined and used to solve for the properties. The effect of aerodynamics on the vehicle will give a good impression in term of performance, cooling system, comfort, stability and visibility. The application aerodynamic in design system will benefit such as improve a fuel consumption, stability vehicle under crosswind, increase performance especially in performing cornering, reduce emission, maximize acceleration and naturally created a cooling system of the engine. For the student formula car competition, to produce down force is essential to improve traction and thus cornering ability.

Eimarace competition was a platform where student would learn and apply a new knowledge especially in physic. The benefits when students join this kind of competition programs, student would construct a prototype of race cars from ground base. The student will inspire with working as a team work that will test the capability especially in communication between top management to the lower level, do branding the team name, collaboration with research universities and explore any organization for purpose sponsorship. These efforts aim on exposure to motorsports. Whenever student start to design, the main factor playing in important role is focusing on aerodynamic drag force (Cd) thus investigate the generation coefficient of lift force (Cl). Although, determination of behavior region, i.e.: wake region, vortex flow, turbulence flow, low pressure, high pressure, down force and separated flow.

In student formula car, the main consideration is to reduce a value of drag force. The drag force remains forces acting in opposite to the relative motion of any object moving with respect to a surrounding fluid. When the fluid flow over the surface, the surface will resist its motion. This is called drag force. Aerodynamic drag is a sum of viscosity and pressure drag. The pressure drag occurs at the two layers of fluid and cause shear force acting. Aerodynamics force and moments analysis especially the drag component is a major interest in several applications including aircraft aerodynamics [1-3], Ferro hydrodynamics [4-5] marine engineering [6] and particularly in automotive industries [7-10]. The aerodynamic drag force of student formula car Society of Automotive Engineer (SAE) was taken into account after the effort of improving the design of the chassis. The design more tend to aerodynamics range that become important not only to improve the acceleration, but also the cooling system at engine side which highly in concern. In order to increase the acceleration, the concentration of down force was highlighted as a stream factor to improve the body of student formula car nearest into ground base. The near body to the ground base, the more tends to reduce the value of Cl and makes the student formula car stable and allow to perform especially in stability and concerning.

ARPN Journal of Engineering and Applied Sciences

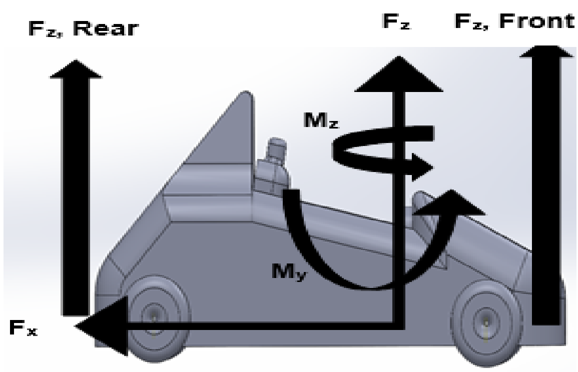
This paper presents a parametric analysis that investigates by Kolej Komuniti Bandar Darul Aman (KKBD) racing team student formula car, whether situated in the range of aerodynamics. This from the numerical investigation result also could expose to determine by undesired region, which identified that could improve for further design improvement in the next application design. Furthermore, from the numerical result will reveal the Cl and Cd where the low pressure and high pressure region, which generated will produce a big impact into acceleration of KKBD student formula car.

Purpose of the study

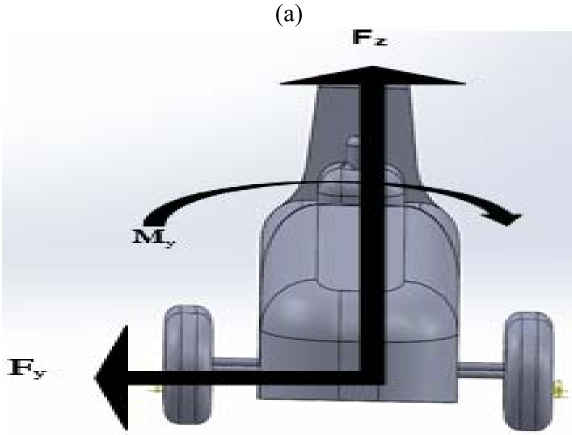
The undesired drag force would affect the performance of the vehicle, reduce acceleration, generate a noise disturbance, giving inconsistence stability, consume more fuel and produce much emission. The important part is to cause the unexpected accident due to swivel lost in control due to the refusal wind. In order to counter the entire problem above, the understanding and the study of aerodynamics is becoming necessary in term of safety and increase the credibility of the design concept.

METHODOLOGY

The aerodynamic force acting in the entire formula student car was defined as:



(a)



(b)

Figure-1. The aerodynamic force acting on KKBD student formula car (a) side view (b) frontal view.

The reaction force acting can be described as below in six sections:

$$F = (Fx; Fy; Fz; My; Mx; Mz) \quad (1)$$

where F_x is a force of drag which opposite from direction of movement or tunnel, F_y is a lateral force and acting in horizontal force which directed to the right hand side, F_z is a force of lift and normally acting upward vertical force, M_x is a rolling momentum and known as momentum of the aerodynamic force reaction in x-axis, M_y is a pitching momentum where momentum of aerodynamic force reaction at y-axis and M_z is a yawing momentum where momentum of aerodynamic forces reaction at z-axis. The front and rear lift force convinces in each rolling train:

$$F_Z = F_{z,} \text{ Front} + F_{z,} \text{ Rear} \quad (2)$$

Taking the coefficient indifferent component:

$$C_x = \frac{2F_x}{p.S.v^2}\,;\; C_y = \frac{2F_y}{p.s.v^2}\,;\; C_z = \frac{2F_z}{p.s.v^2} \quad (3)$$

$$C_l = \frac{2.M_x}{p.S.l.v^2};\; C_m = \frac{2.M_y}{p.S.l.V^2};\; C_n = \frac{2.M_z}{p.S.l.v^2} \quad (3)$$

where S is the frontal area of the vehicle, l is a characteristic of vehicle length, p is the air density, V represents the air velocity and F_c indicates the cooling aerodynamic drag which defined as the difference of drag in vehicle production. In the virtual wind tunnel, the wind tunnel simulator was developed in ANSYS version 14.

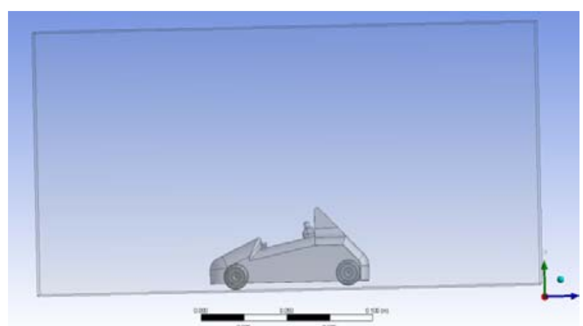


Figure-2. KKBD car in virtual wind tunnel.

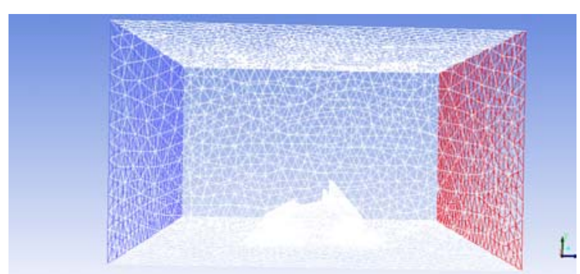


Figure-3. Ultra-meshing generation.

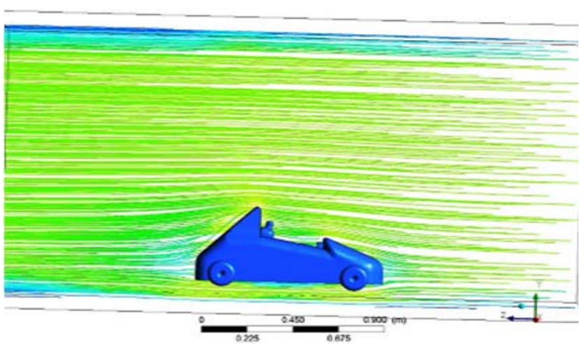
VOL. 11, NO. 20, OCTOBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

Wind tunnel was designed and the test section is 5m wide and 3m high. The maximum speed was set at 27.22m/s or equal to 100km/h. The wind tunnel was set especially in ground floor, tyre contact is approximately 0.001m, the length between the body and the inlet region around 5m length, the body and outlet length which approximately 7m.

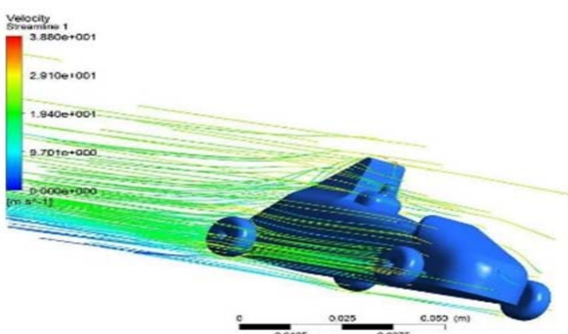
Air is blown in a high velocity through a duct which equipped with a viewing port and instrumentation where a model was mounted to perform the study. The main concern in this virtual wind tunnel is to expose the separated flow that might raise the value of Cd. The main inlet, outlet and wall were set as the boundary layer.

RESULTS AND DISCUSSION

Effect of streamline velocity



(a)



(b)

Figure-4. Streamline result (a) frontal view (b) side view.

Figure-4 illustrates the prediction of streamlining whenever the KKBD student formula car in 100km/h acceleration. The red line represents the stressed line in high velocity which encounter obstacle and the area stressed that focusing the main roll hoop firewall, driver helmet, front tyre and rear tyre. The green and yellow signify the flow normal thru blond body and the separated flow occur in the rear between the top main roll hoop and engine cover compartment. Separated flow also occurred at bottom rear engine compartment cover. For the further development study, the slope of diffuser such as sedan car performance essentially to be applied in KKBD student formula car to improve the separated flow.

Top view streamline

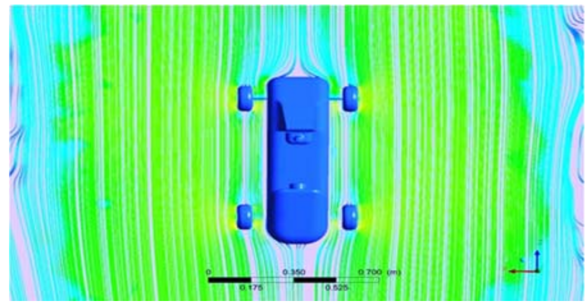
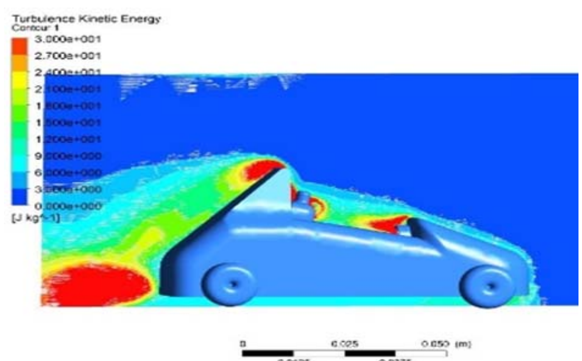


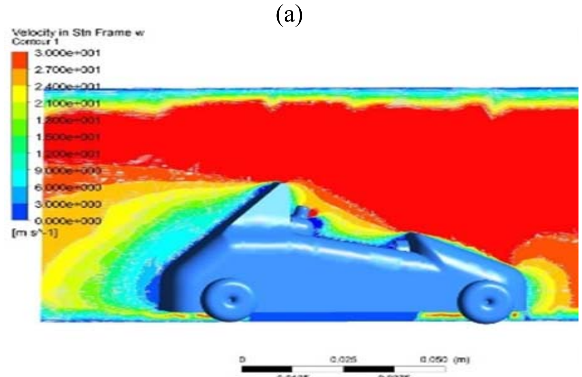
Figure-5. Top view streamline.

Figure-5 of top view shows the unattached flow (separated flow at the rear) that has been occurring. Despite separated flow has a minor area at the rear side, it could tend to generate turbulence flow thus made low pressure at rear side which contribute to increase the Cd. Another region was tyre. But in this case, turbulence flow aid to the cooling system for disc brake and useful to reduce undesired heat which produced during braking operation.

Effect of contour



(a)



(b)

Figure-6. Contour region (a) turbulence kinetic energy (b) velocity in the 5th frame (W).

Figure-6 shows the prediction of the contour pattern region. From the result indicates that the tremendous effect on the way a car accelerate in the laminar air flow. The main concern on this section is wake region. Wake region is the region of re circulating flow immediately behind a moving or stationary in the formula student car. Nevertheless, the area that wake region restricted merely at dashboard, main roll hoop firewall and rear end engine cover compartment. In spite of wake region, from the resultant velocity in the 5th frame (W), it reveals the low pressure which occurred in the bottom of line ground body. This shows a good sign in aerodynamic where high pressure acting on the top body of KKBD car, thus low pressure occurred at bottom side that would escalate a stability and cornering efficiency.

Results of coefficient of drag (Cd)

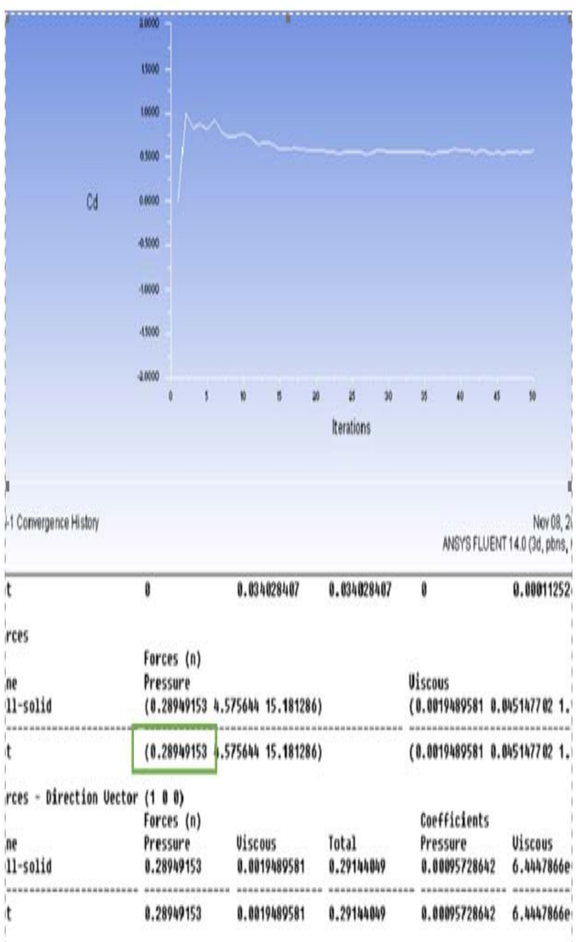


Figure-7. Coefficient of drag (Cd).

Figure-7 shows the simulation result which performed by Fluent CFD solver, where the Cd = 0.28. From the Cd value may determine that the KKBD student formula car was in range of aerodynamic specification and have a tendency to perform in the race. Furthermore, the coefficient of lift value is a low level adaptation thus increase a confidence level of the driver when performed in cornering.

CONCLUSIONS

Numerical study of aerodynamic drag force on student formula car has been made as well. A virtual simulation study has brought a result where inspired student to increase a confident level, especially in the design concept of student formula car that would suggest the best design in future development. From the simulation study also can observe the area which generated turbulence flow, low pressure, high pressure, coefficient of drag, the coefficient of lift, wake area, pressure distribution on the surface, trailing vortices flow, noise study, skin friction drag and pressure drag. In the simulation study revealed that Cd = 0.28 has a tendency to produce aerodynamic student formula car range. Furthermore, from the simulation study also found that the separated flow occurs whenever the student formula car moving above 50km/h. The more blunt design the vehicle, the higher coefficient of drag will produce.

ACKNOWLEDGEMENTS

We would like to express deepest thanks to the Terengganu Advance Technical Institute (TATIUC), Universiti Sains Malaysia and Kolej Komuniti Bandar DarulAman for facilities support in this research progress journey.

REFERENCES

[1] M. H. Djavareshkian, A. Esmaeli and A. Parsani. 2011. Aerodynamics of smart flap underground effect. Aerospace Science and Technology. 15(8): 642-652.

[2] K. V. Rozhdestvensky and V. A. Ryzhov.2003. Aerohydrodynamics of flapping-wing propulsors. Progress in Aerospace Sciences. 39(8): 585-633.

[3] P. R. Viswanath. 2002. Aircraft viscous drag reduction using riblets. Progress in Aerospace Sciences. 38(6): 571-600.

[4] C. Baker.2010. The flow around high speed trains. Journal of Wind Engineering and Industrial Aerodynamics. 98(6): 277-298.

[5] R. S. Raghunathan, H. D. Kim and T. Setoguchi.2002. Aerodynamics of high-speed railway train. Progress in Aerospace Sciences. 38(6): 469-514.

[6] A. H. Day and C. Cooper. 2011. An experimental study of interceptors for drag reduction on high-performance sailing yachts. Ocean Engineering. 38(8): 983-994.

[7] S. Watkins and G. Vino. 2008. The effect of vehicle spacing on the aerodynamics of a representative car

shape. Journal of Wind Engineering and Industrial Aero-Dynamics. 96(6): 1232-1239.

[8] S. Y. Cheng, M. Tsubokura, T. Nakashima, T. Nouzama and Y. Okada. 2011. A numerical analysis of transient flow past road vehicles subjected to pitching oscillation. Journal of Wind Engineering and Industrial Aerodynamics. 99(5): 511-522.

[9] S. A. A. A. Ghani, A. Aroussi and E. Rice. 2001. Simulation of road vehicle natural environment in a climatic wind tunnel. Simulation Practice and Theory. 8(6): 359-375.

[10] A. Gharibi and R. Khaki. 2015. A method for calculating aircraft aerodynamic loads using obtained data from flight simulation. Indian Journal of Science and Technology. 8(S7): 74-84.