Journal of Research and Applications in Mechanical Engineering
ISSN: 2229-2152 (Print); 2697-424x (Online)
(2022) Vol. 10, No. 1, Paper No. JRAME-22-10-001
DOI: 10.14456/jrame.2022.1

Research Article

NUMERICAL STUDY AND DESIGN OF SIDE-DIFFUSERS FOR FORMULA-SAE CAR

C. Chatpattanasiri
K. Bunpapong
M. Sahastharachai
B. Puangkird*
Department of Mechanical Engineering,
Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, Thailand

Received 20 November 2021
Revised 23 January 2022
Accepted 28 January 2022

ABSTRACT:
This research focuses on designing side-diffusers for Formula Student race car. The aim is to reduce the lap time in a circular track called Skid-pad event by mean of increasing downforce generated by the diffusers, a ground effect aerodynamic device. Utilizing three-dimensional CFD simulation via cloud computing to obtain the most optimal design among the studied parameters. The study employs SST k-omega turbulence model with an incompressible flow and isothermal assumptions and the CFD model is validated via wind tunnel testing. Finally, the results show that the optimal outlet angle of 32 degrees due to the behavior vortex above the diffuser and the higher the outlet to inlet height, the lesser downforce.

Keywords: *CFD, Formula Student, Diffusers, Skid-pad, SST k-omega, Downforce*

1. INTRODUCTION

Performance of race cars can be viewed in various ways including cornering capability. According to Newton's laws of motion, it is known that the cornering speed increase when the downforce increase. From the mid-1960s until the present time, aerodynamic devices such as front and rear wings have been widely utilized and improved in order to maximize the downforce while minimizing drag [1-5].

In 1970, the Chaparral 2J race car generated up to 1.5 times the downforce of its own weight by installing a large fan to suck the air under the car together with the construction of a side air trap device called "Skirts" employing the ground effect to create unusual downforce for the car in the Can-Am race [6]. This method is later considered to be illegal for the employment of an active aerodynamic device, the aerodynamic device which requires an external energy source.

In 1978, Colin Chapman and Lotus's team, once again, implemented skirts and underbody design of diffuser shape with the floor as the bottom wall of diffuser to take advantage of the ground effect [7] making the car produced more downforce than every other team resulting in increased cornering speed and hugely reduced braking distance. The diffuser shape and skirts became popular among race car teams for years until the FIA decided to forbid the use of movable skirts that always make contact to the ground in 1981. The use of underbody design of diffuser shape was later banned and replaced with flat underbody in 1983. Despite the above regulations, ground effect still plays a big role on designing aerodynamics aspect of race cars and receives great deals of attention from researchers [8-11].

* Corresponding author: B. Puangkird
E-mail address: bumroong.pu@kmitl.ac.th

Many researches have been made to study the shape of the diffuser at the back edge of the race car's underbody [12-14]. However, there are very few about side-diffusers that attached to the spaces between the front wheel and rear wheel. This research focuses on employing computational fluid dynamics (CFD) simulation to study parameters that affect the performance of side-diffusers and design it for undergrad's automotive club at King Mongkut's Institute of Technology Ladkrabang (KMITL) which has always been participating in FSAE competition in Thailand.

2. OBJECTIVE DETERMINATION

The side-diffusers is to be design for KF2018/2019 which has the properties as shown in Table 1.

Table 1: The properties of KF2018/2019

Mass	266	kg
Tire friction coefficient (μ)	1.30	-
Cross-sectional area	1.00	m^2
Power	41.0	kW

In order to study and improve cornering capability of the car, this research focuses on Skid-pad event which has a circular track as shown in Fig. 1.

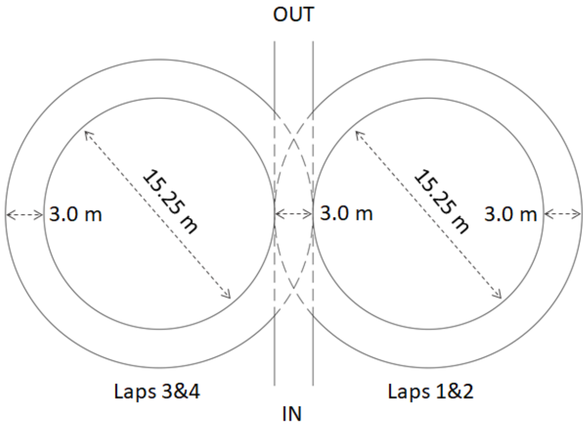


Fig. 1. The track for Skid-pad event

The target time is set to be 4.900 s per lab in order to beat the car with the best record from last year competition which is 4.951 s per lab from Shibaura Institute of Technology.

For the sake of simplicity, the motion of the car in Skid-pad event is modeled to be an object in circular motion. When the car's speed reaches the maximum point, the motion of the car can be described by Equation (1).

$$\mu(mg + F_{down}) = m\frac{v^2}{R} \tag{1}$$

The Radius (R) is taken to be 8.325 m corresponding to maximum speed of 10.68 m/s and use gravitational acceleration (g) of 9.81 m/s^2. From Equation (1), the downforce required is calculated to be 196 N or lift coefficient of 2.81 (direction downward) which become the design target. Without diffusers, the car with front and rear wings generates the downforce of 142 N at 10.68 m/s or 40.0 km/h. The remaining 54 N is to be achieved by mean of side-diffusers which is designed by CFD simulation using SimScale, an OpenFOAM based cloud computing software which perform CFD simulation using finite volume method.

3. VALIDATION

3.1 Wind tunnel

The results from CFD simulation is to be validated by the experimental results from wind tunnel testing. Figure 2 shows the wind tunnel used in this research owned by Mechanical Engineering Department, KMITL.

Fig. 2. Wind tunnel

The reliability of the wind tunnel has been tested by conducting experiment of cylinder external flow using PVC pipes, experiment 1 with a pipe with 22 mm diameter while experiment 2 with a pipe of 34 mm diameter. The two pipes have the same length of 280 mm. The results are shown in Fig. 3.

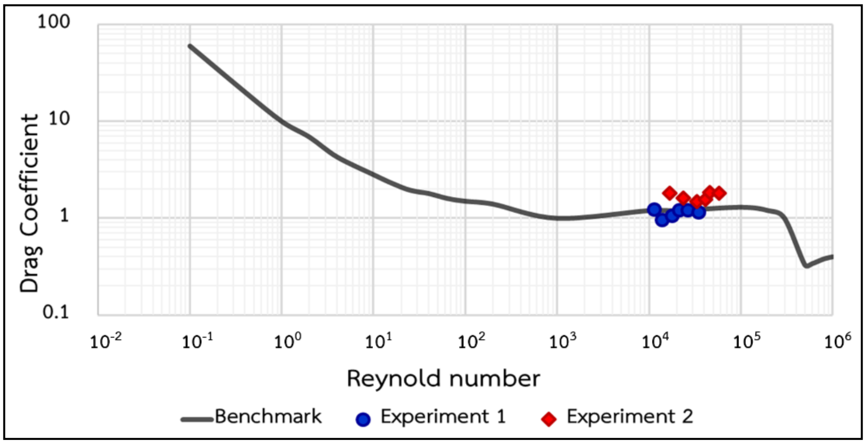


Fig. 3. Results from the experiment of cylinder external flow

The benchmark in Fig. 3 is from a similar experiment conducted by Tritton [15] which concerns infinitely long cylinder. It is clear that the results from experiment 1 are closer to the benchmark than the results from experiment 2. This is because the length-to-diameter ratio of the cylinder from experiment 1 is bigger than the same ratio from experiment 2. Therefore, the flow pass through the cylinder from experiment 1 is more similar to the benchmark. The results also show that both experiments are sufficiently close to the benchmark. Thus, the reliability of the wind tunnel is confirmed.

3.2 Validation of flow through a diffuser

The validation is done by conducting a CFD simulation of air flow through an arbitrary shape of a diffuser and compare the results to the wind tunnel experiment results of the same shape. CFD simulations in this research use Reynolds-averaged Navier-Stokes (RANS) equations with *SST k-omega* turbulence model.

Firstly, mesh independent test is done in order to know how fine mesh should be when dealing with CFD simulation of diffusers. This process also ensures that computing resources are being used efficiently. The results from the mesh independent test are in Fig. 4.

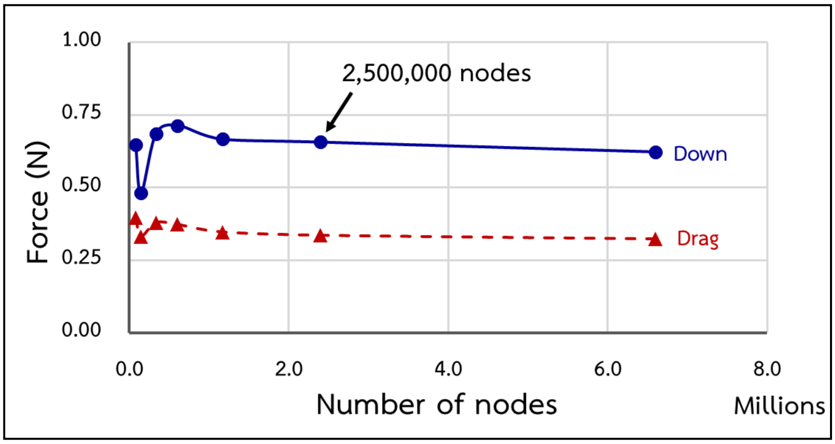
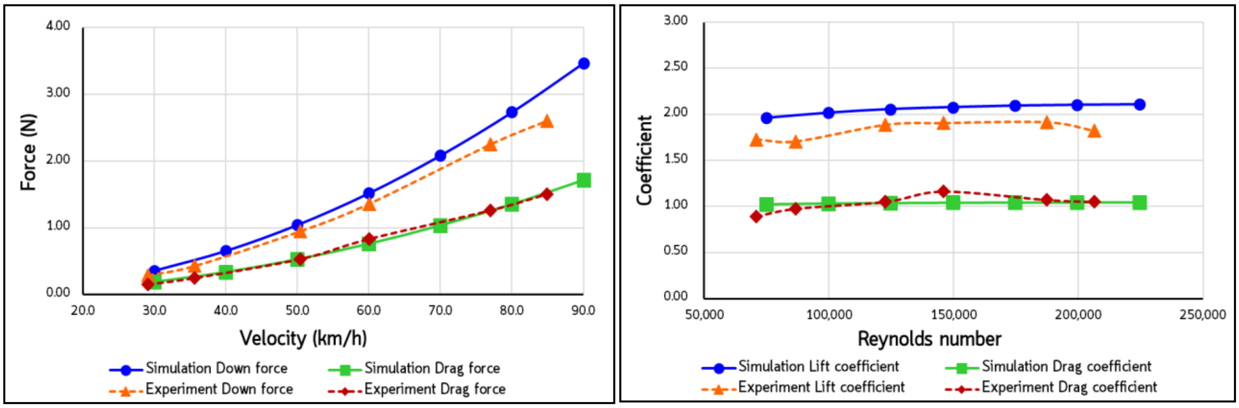


Fig. 4. Mesh independent testing of flow through a diffuser

Figure 4 shows that the results become independent of the number of nodes at approximately 2,000,000 nodes. The same mesh is used to do CFD simulation between 30 to 90 km/h. The simulation results are compared to the results from wind tunnel experiment in Fig. 5 a) and b). Note that the Reynolds number is calculated using the diffuser length as characteristic length.



a) b)

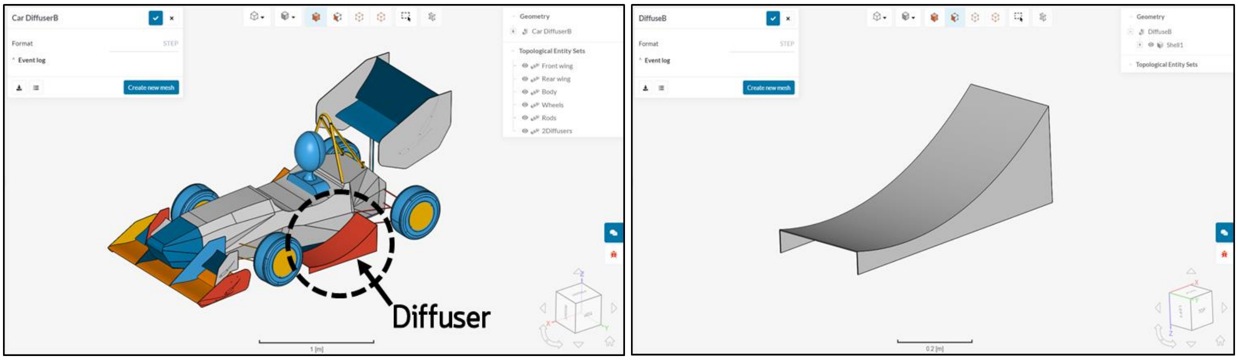
Fig. 5. Comparison of the simulation results and the experimental results a) presented forces and velocity, b) presented force coefficient and Reynolds number

Both figures represent the same results but presented in two different forms, dimensional and dimensionless. They confirm that the CFD simulation and the wind tunnel experiment give acceptably similar results.

4. DESIGN PROCESS

4.1 The choice of simulation apart from the car or attached to the car

CFD simulation of diffuser can be done in 2 different methods, apart from the car and attached to the car, as shown in Fig. 6. Noted that all the simulations involved with the whole race car in this research are done without rotating wheels boundary condition (the wheels are fixed as part of the rigid body representation of the frame.) This simplification has been justified in the work done by S Shao et al. [16].



a) b)

Fig. 6. Simulation a) attached to the car and b) apart from the car

It is preferable to do the CFD simulation of diffusers apart from the car because less computing resources are required. However, this method involves great simplification of the geometry domain which produce additional error. To prove that it is still acceptable to do the simulation apart from the car, simulations of 3 arbitrary diffusers is completed to compare the results of both methods. The comparison is shown in Fig. 7. The results show similarity of both methods of simulations. Therefore, the apart-from-the-car method is used in this research.

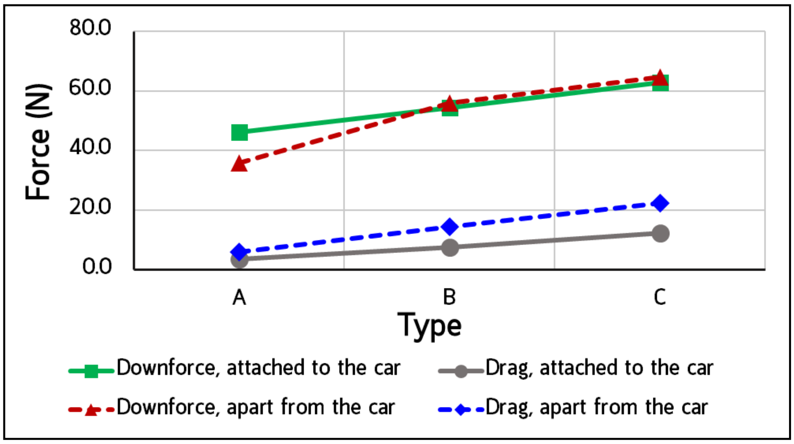


Fig. 7. Comparison of the simulation results apart from the car and attached to the car

4.2 The design process of the diffuser is divided into 3 steps

4.2.1 Design step 1

Shapes of simple flat diffusers, Fig. 8, is studied by varying only back height (h_3). For other parameters, diffuser's length (L) and width (W) are 75 mm and 300 mm due to the car regulation while front height (h_1) is 10 mm for the sake of simplicity.

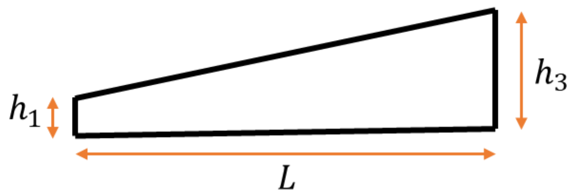


Fig. 8. Diffusers in step 1

The CFD simulation results, Fig. 9, shows that while the back height (h_3) increases, the downforce increases. It is not practical to have back height (h_3) exceeded 350 mm due to limitation of space in the real car. Therefore, back height (h_3) is set to be 350 mm to obtain the maximum downforce of 17.1 N.

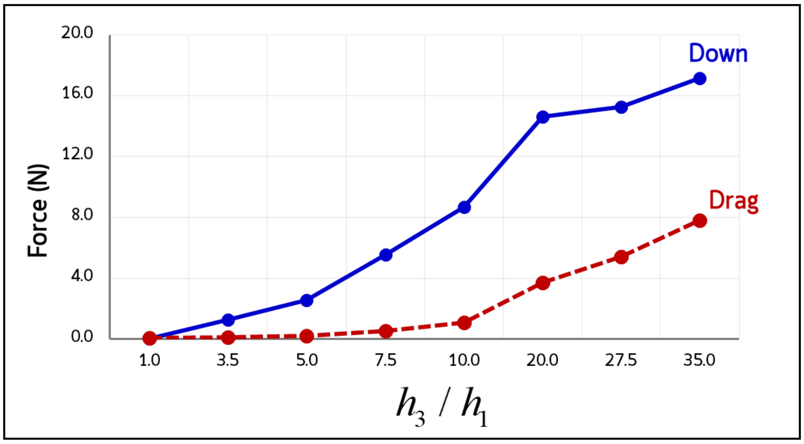


Fig. 9. Relationship between forces and h_3/h_1 from CFD simulation

4.2.2 Design step 2

In step 2, the shapes of diffusers are further optimized by varying both the back height to front height ratio (h_3/h_1) and the diffuser's angle (α), shown in Fig. 10. The ratio is varied by simply varying the front height (h_1). The diffuser's angle (α) is varied by changing the horizontal distance of the critical point. Note that the critical point's height (h_2) is set to equal to the front height (h_1) and the back height (h_3) remains constant of 350 mm.

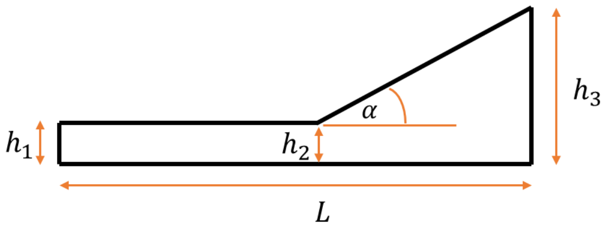


Fig. 10. Diffuser in step 2

The simulation results are shown in Fig. 11 with each line represents each back height to front height ratio (h_3/h_1). The results show that as the ratio (h_3/h_1) increase, downforce decrease. In this case, the smallest ratio (h_3/h_1) available is 3.5 otherwise the front height (h_1) would not be able to fit in the allowable space of real car. For diffuser's angle (α), the maximum down force is at 32 degree. Thus, the ratio (h_3/h_1) and the diffuser's angle (α) is set to 3.5 and 32 degree. The downforce produced is 30.0 N.

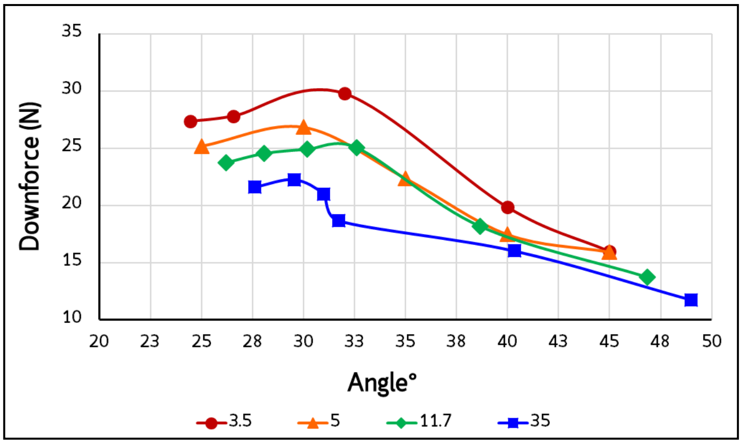


Fig. 11. Relationship between the downforce and diffuser's angle of each ratio, For flat diffuser

4.2.3 Design step 3

In the final step, all the studied parameter remains the same as in step 2. The diffuser is changed from flat type to curved type using 3 points spline of front edge, back edge and critical point as shown in Fig. 12.

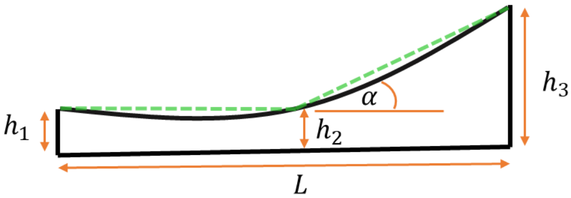


Fig. 12. Diffuser in step 3

The results from CFD simulations of diffusers in step 3 is shown in Fig. 13. The maximum downforce is 34.4 N which occurs at the ratio (h_3/h_1) and the diffuser's angle (α) of 3.5 and 32 degree, same as in step 2. When attached to the car, left and right together produce the downforce of 68.8 N. This diffuser is the final design in this research.

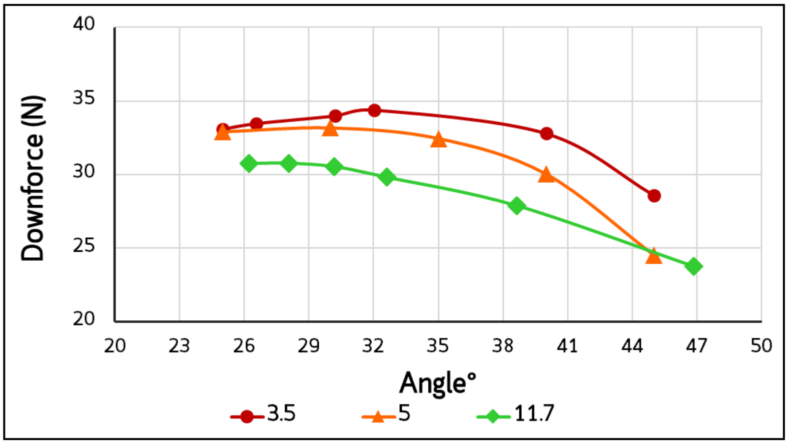


Fig. 13. Relationship between the downforce and diffuser's angle of each ratio for curved diffuser

5. RESULTS AND DISCUSSION

5.1 Performance of the race car with diffusers

Table 2 Shows the Aerodynamic performance of KF2018/2019 with and without the final design of diffuser, 32-degree spline diffuser, attached to left and right sides of the race car. These results are also obtained by CFD simulation, but this time using attached-to-the-car method.

Table 2: KF2018/2019 with and without the final design of diffuser

	Without diffusers	With diffuser
Downforce (N)	142	207
Drag (N)	90.0	101
Lift coefficient	2.04	2.98
Drag coefficient	1.29	1.45

The race car with diffuser can produce the downforce of 207 N, corresponding to lift coefficient of 2.98 at velocity of 10.68 m/s. This is more than the target at 196 N or lift coefficient of 2.81 meaning that the KF2018/2019 can achieve the target time of 4.90 s per lab in Skid-pad event. Noted that the side-diffusers generates as much as 30% of the overall downforce while only producing only 10% of drag. Moreover, the estimated down-drag ratio of side diffuser can be calculated to be 5.90 which is much higher than that of combined front and rear wings which is roughly 3.25 (the performance data of front and rear wings is not shown for brevity.) This shows that side-diffusers is an effective aerodynamic device.

5.2 Investigation of the flow characteristic inside diffusers

CFD simulation results from SimScale can be visualized by ParaView. Figure 14 shows the flow of air through 3 different models of the diffuser, presenting the velocity magnitude and pressure. The 3 diffusers are;

1. 32-degree flat diffuser (Top) represents diffusers that have optimal diffuser's angle (α) but using flat type.
2. 45-degree spline diffuser (Middle) represents diffusers that have non-optimal diffuser's angle (α).
3. 32-degree spline diffuser (Bottom) as the final design to compare with the other two models.

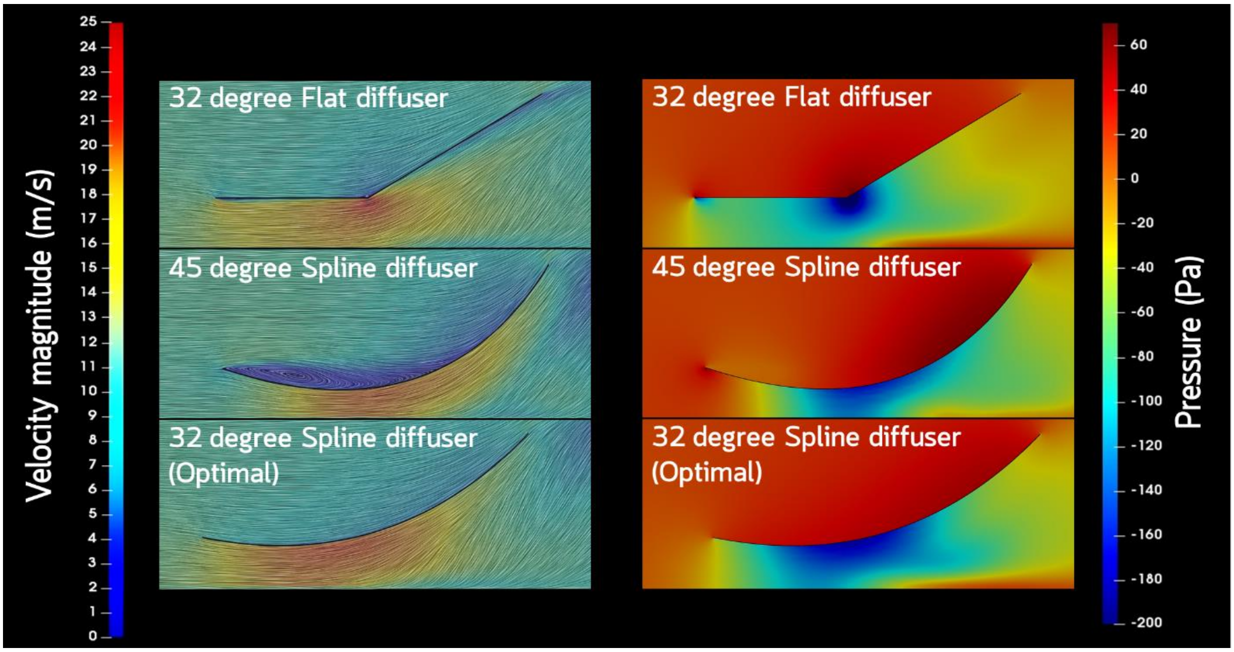


Fig. 14. Visualization of air flow through 3 different models of diffusers

Downforce is the resultant force calculated from the integration of pressure on the area of the top surface of a diffuser. Higher pressure difference means higher resultant downforce. From the basic Bernoulli principle, pressure increases when velocity decreases, and vice versa. This principle only applies when the flow is not very turbulent, meaning that it is not true in the area of circulation or vortex.

Figure 14 confirms that 32-degree spline diffuser has the maximum downforce among the three diffusers. Both the high-pressure-zone and low-pressure-zone spread widest across the top surface of the diffuser.

For 45-degree spline diffuser the vortex above the diffuser's top surface reduce the size of high-pressure-zone as shown in Fig. 15. Also, the low-pressure-zone does not spread as wind as in 32-degree spline diffuser. Thus, this model generates less downforce.

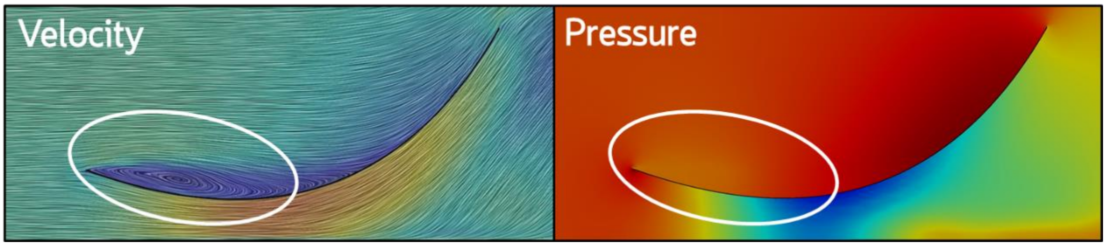


Fig. 15. Vortex occurs above the top surface of 45-degree spline diffuser

For 32-degree flat diffuser, the sudden change of angle at the critical point causes the low-pressure-zone to be relatively small. Figure 16 also shows a small vortex above the diffuser's top surface, at the critical point, which reduces pressure at the high-pressure-zone above the diffuser. Therefore, this model generates less downforce than the final design as well.

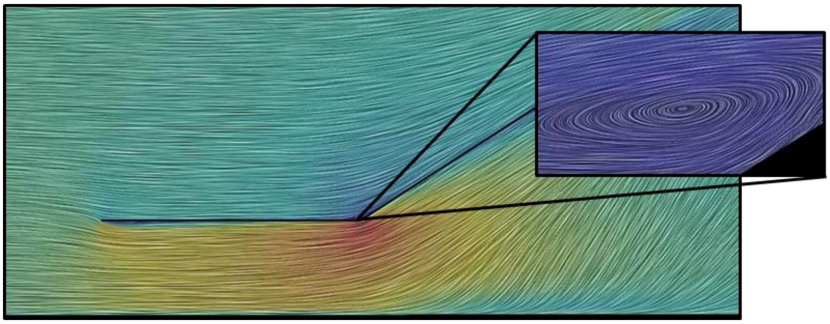


Fig. 16. Vortex occurs above the top surface of 32-degree flat diffuser

6. Conclusion

The shapes of side-diffusers have been numerically studied and designed by CFD simulations validated by wind tunnel experiments. The CFD results are in agreement with the results from the experiments. It has been confirmed that CFD results of side-diffusers both apart-from-the-car method and attached-to-the-car method give acceptably similar results. Therefore, CFD simulations of side-diffuser can be done using apart-from-the-car method.

For back height (h_3) = 0-350 mm and front height (h_1) = 0-100 mm, downforce generated by diffusers increase with the increase of back height and the decrease of back height to front height ratio. Curved diffuses generate more downforce compare to flat diffuser with the same dimensions and diffuser's angle. The optimal diffuser's angle (α) is found at 32 degree for both flat diffusers and curved diffusers because diffusers with the angle of 32 degree do not generate vortex above its top surface.

When attached to the race car, the optimal diffuser found (shown in 4.4) contributes as much as 30% of the overall downforce while generating only 10% additional drag. This makes the car reach the downforce of 207 N which corresponds to the lift coefficient of 2.98 which is even more than the set target of 196 N.

7. ACKNOWLEDGEMENT

This work could not be accomplished without helpful suggestions of the adviser Dr. Bumroong Puangkird, and every stuff from the Department of mechanical engineering, KMITL.

NOMENCLATURE

μ	Tire friction coefficient
m	Mass, kg
g	Gravitational acceleration, m/s^2
F_{down}	Downforce, N
v	velocity magnitude, m/s
R	Radius, m
h_1	Diffuser's front height, m
h_2	Diffuser's critical point height, m
h_3	Diffuser's back height, m
L	Diffuser's length, m
W	Diffuser's width, m
α	Diffuser's angle, degree

REFERENCES

[1] Katz, J. Race car aerodynamics: Designing for speed, 1995, Bentley, Cambridge.

[2] Wordley, S. and Saunders, J. Aerodynamics for formula SAE: Initial design and performance prediction, SAE Technical Paper, 2006, pp. 1-12.

[3] Wordley, S. and Saunders, J. Aerodynamics for formula SAE: A numerical, wind tunnel and on-track study, SAE Technical Paper, 2006, pp. 1-15.

[4] Hetawal, S., Gophane, M., Ajay, B. and Mukkamala Y. Aerodynamic study of formula SAE car, Procedia Engineering, Vol. 97, 2014, pp. 1198-1207.

[5] Li, C., Li, J., Wu ,N. and Zhang, S. Optimization design of FSAE racer body based on CFD, Journal of Machine Design, Vol. 31(8), 2014, pp. 70-73.

[6] Falconer, R. and Nye, D. Chaparral: Complete history of Jim Hall's Chaparral race cars 1961-1970, 1992, Motorbooks International, Richmond.

[7] Reid, E.G. A full-scale investigation of ground effect, 1927, Washington, Government printing office.

[8] Ranzenbach, R. and Barlow, J.B. Two-dimensional airfoil in ground effect, an experimental and computational study, SAE Technical Paper, 1994, pp. 1-9.

[9] Jasinski, W.J. and Selig, M.S. Experimental study of open-wheel race-car front wings, SAE Technical Paper, 1998, pp. 2549-2557.

[10] Zerihan, J. and Zhang, X. Aerodynamics of a single element wing in ground effect, Journal of Aircraft, Vol. 37(6), 2000, pp. 1058-1064.

[11] Zhang, X. and Zerihan, J. Aerodynamics of a double-element wing in ground effect, AIAA Journal, Vol. 41(6), 2003, pp. 1007-1016.

[12] Cooper, K.R., Bertenyi, T., Dutil, G., Syms, J. and Sovran, G. The aerodynamic performance of automotive underbody diffusers, SAE Technical Paper, 1998, pp. 150-179.

[13] Ruhrmann, A. and Zhang, X. Influence of diffuser angle on a bluff body in ground effect, Journal of fluids engineering, Vol. 125(2), 2003, pp. 332-338.

[14] Jowsey, L. and Passmore, M. Experimental study of multiple-channel automotive underbody diffusers, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, Vol. 224(7), 2010, pp. 865-879.

[15] Tritton, D.J. Experiments on the flow past a circular cylinder at low Reynolds numbers, Journal of Fluid Mechanics, Vol. 6(4), 1959, pp. 547-567.
[16] Shao, S., Zhang, Y., Zhao, J. and Tang, W. The influence of wheel rotating to FSAE racing car aerodynamic characteristics, Applied Mechanics and Materials, Vol. 300-301, 2013, pp. 1054-1057.