Computational fluid dynamics (CFD) analysis of the divergent part of the Formula Student car restrictor

Marko Lučić
Faculty of Mechanical Engineering, University of Montenegro, Podgorica, Montenegro
E-mail: markol@ucg.ac.me

Abstract: *The FSAE competition is a student engineering competition that involves designing, constructing, and building a small racing car. Students from different universities compete on auto-moto sports tracks. The competition's judges are eminent experts in marketing, automotive engineering, and racing car engineering. In class IC, engines are used as power units. One of the main limitations is that all intake air must pass through a diameter of 20 mm. One of the main challenges facing student competitors is solving this problem, and the convergent-divergent jet is one of the possible solutions to the problem. In this paper, CFD simulations were used to examine the influence of the divergent part of the restrictor on the total mass flow at the nozzle exit. A diagram of the dependence of the mass flow on the half-angle of the divergent part was obtained. For the CFD simulation, ANSYS Fluent was used, which proved to be very good for examining the mentioned influence.*
Keywords*: CFD, VEHICLE, RESTRICTOR, FORMULA STUDENT, INTAKE MANIFOLD*

1. Introduction

Computational Fluid Dynamics or CFD for short is the analysis of systems involving fluid flow, heat transfer, and chemical reactions using computer simulation. This technique is very powerful and covers a wide range of industrial and non-industrial applications. Since the 1960s, the aerospace industry has integrated CFD techniques into the research, development, design, and production of aircraft and jet engines. After that, these methods are applied to the design of internal combustion engines, gas furnaces, and turbines. Nowadays, vehicle manufacturers routinely predict drag forces, and airflow under and around cars using CFD. Nowadays, CFD is one of the main components in the design of industrial products. One of the main reasons why CFD has lagged is the great complexity of the underlying behavior, which makes it impossible to describe fluid flows. As the availability of affordable high-performance hardware grew, the introduction of user-friendly interfaces led to an increase in interest in CFD. Only since the 1990s has CFD entered the wider industrial community. CFD codes are structured around numerical algorithms that can deal with fluid flow problems. To provide easy access to their solving power, all commercial CFD packages include a simple user interface for entering problem parameters and examining results [1]. The application of CFD simulation in the Formula Student competition is very large, as evidenced by the large number of papers [2-17] based on CFD techniques.

This work is divided into six chapters. The first chapter is related to the introduction, while the second is related to the limitations when designing the intake system of the Formula Student vehicle. The third chapter presents the CFD simulation of the restrictor, while the fourth presents the results of the CFD simulation. The fifth chapter is related to the conclusions reached during the research, while the last sixth chapter contains a list of references used in this work.

2. Formula Student restrictor

The restrictor is an element that according to the Formula Student Rules [18] the engine that will drive the vehicle in class IC FSAE competition is limited to 710 cm3. Also, all the air that is sucked into these engines has to go through the restrictor. There are two possible configurations of the air intake system that are allowed by the competition organizers. The configuration on naturally aspirated engines is shown in Figure 1 and the configuration with turbocharged or supercharged engines is shown in Figure 2.

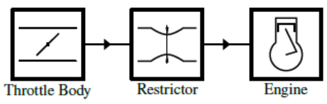

Fig. 1 *Configuration for naturally aspirated engines [18]*

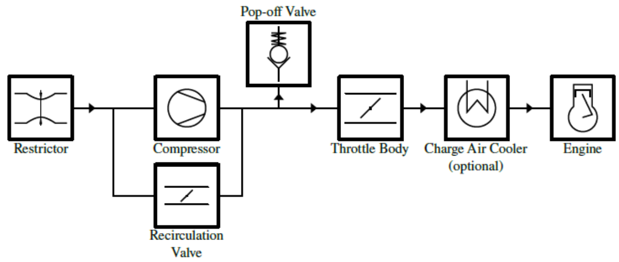

Fig. 2 *Configuration for turbocharged or supercharged engines [18]*

It can be seen that the configuration from Figure 1 is significantly simpler than the configuration from Figure 2, but the configuration with turbocharged or supercharged engines is significantly more expensive. As a rule, the configuration from Figure 2 should provide a higher airflow, because the compressor enables a higher flow than when there is no airflow. The air intake system is one of the systems that need a lot of design attention. As a rule, if more air is allowed to be sucked into the engine, greater power of the power unit will be obtained. As it is necessary to allow the air that entered the engine to leave it, great attention should also be paid to the design of the exhaust system. The intake and exhaust systems interact with each other through the engine, so if one of these two systems is poorly designed, the potential of the well-designed system will not be extracted. For the design of the intake system to be considered good, it needs to have a good flow characteristic, that is, to enable the best possible intake of air into the engine. Formula Student teams are condemned to a large extent to use engines that are installed in motorcycles due to engine volume limitations. These engines need to be adapted for use on FSAE vehicles, so it is necessary to redesign the intake and exhaust systems as well as some other systems such as the fuel distribution system and the engine cooling system. As these engines, when installed in FSAE vehicles, will work on a different mode than the one they work on in motorcycles, it is necessary to perform a correction of the engine maps. In the last case, the possibility of redesigning parts such as the crankshaft, piston, or connecting road can be considered. A very useful analysis when considering the redesign of these parts can be a dynamic analysis of the piston group. This analysis is preceded by a kinematic analysis, and how it can be done very quickly and efficiently using the CATIA software is shown in the paper [19]. Figure 3 shows the intake system concept of one of the Formula Student teams.

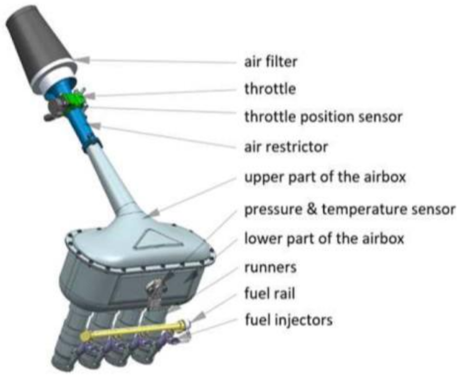

Fig. 3 *Design of the assembly of the intake manifold [10]*

3. CFD simulation of restrictor

As pointed out in the introduction, CFD techniques have become the standard when designing industrial products. Ansys FLUENT software was used for CFD simulation. This chapter presents a CFD simulation of airflow through a restrictor. As the goal of the entire research was to examine the influence of the half-angle restrictor on the quality of the flow, all parameters were constant in each CFD simulation, while the value of the half-angle was changed by 0.5 degrees. Six simulations were made, that is, six fluid domains with different half-angle values were generated. Figure 4 shows the dimensions of the restrictor. From Figure 4, it can be seen that all dimensions are constant except for the half-angle, which is marked with the letter "a". The diameter of the restrictor throat is 20 mm and it is also a constant size. The variable is the half-angle, which is different in each CFD simulation. The half-angle values considered in this analysis are 4.5º, 5.0º, 5.5º, 6.0º, 6.5º, and 7.0º. The constant values are:

- entrance diameter 50 mm,
- length of the convergent part 35 mm,
- the radius of the convergent part 50º,
- length of divergent part 225 mm

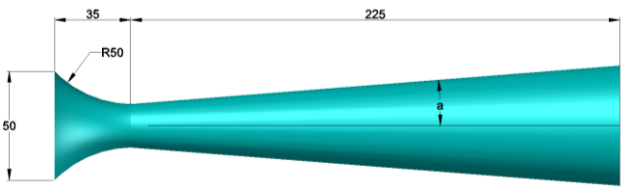

Fig. 4 *Dimension of restrictor*

The geometry (3D geometry) of the fluid domain is shown in the Ansys software in Figure 5. In Figure 6, the network of the fluid domain of the restrictor is shown. Mesh generation is one of the very important steps in CFD simulation.

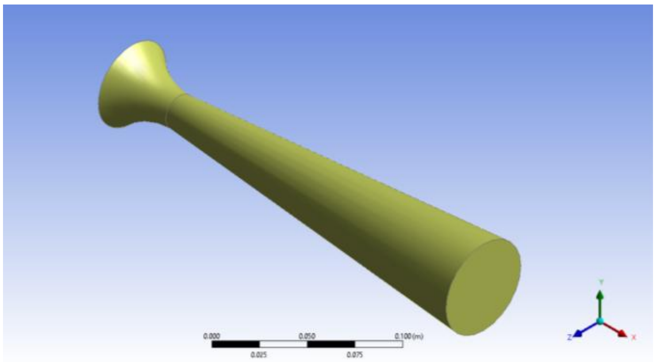

Fig. 5 *3D model fluidnog domena restrictora*

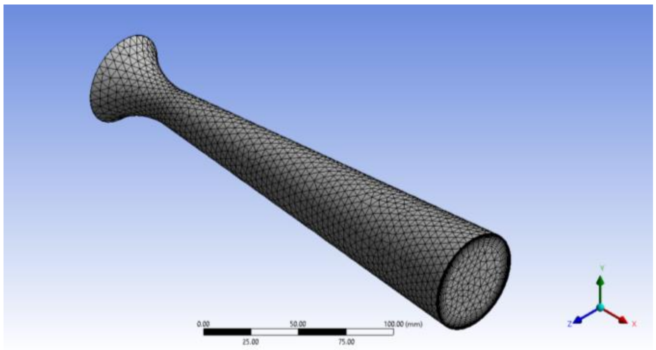

Fig. 6 *Mash of fluid domain restrictor*

When setting up a CFD simulation, one of the most important steps is defining the boundary conditions. To set the boundary conditions in a good way, it is necessary to know the physics of the problem itself. Air was used as the fluid. The flow of air through the restrictor occurs thanks to the pressure difference at the inlet and outlet of the restrictor. This difference amounts to 0.1 bar because an inlet pressure of 1 bar is defined at the inlet and an outlet pressure of 0.9 bar at the outlet. Figure 7 shows the inlet and outlet of the restrictor.

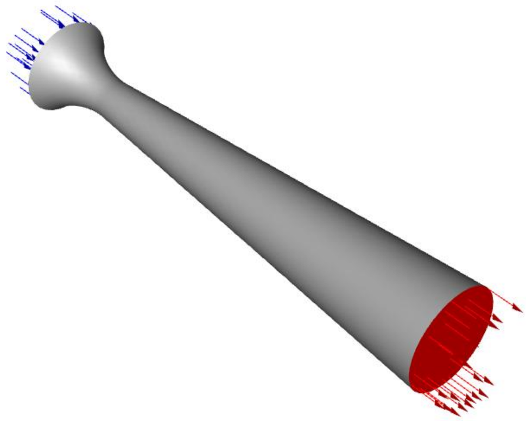

Fig. 7 *Inlet (blue arrows) and outlet (red arrows)*

4. Results

The CFD simulation results are shown in the form of mass flow rate on the restrictor outlet and average axial velocity on the restrictor outlet. These values were determined through the surface double integral and are shown in table 1.

Table 1: *Mass flow and Average axial velocity on restrictor outlet*

Half-angle	Mass flow outlet (kg/s)	Average Axial Velocity outlet (m/s)
4.5	0.13711	46.72823
5	0.13614	40.78013
5.5	0.13487	35.57703
6	0.13310	31.73427
6.5	0.13043	27.88637
7	0.12634	26.47269

The dependence of mass flow on the restrictor outlet when changing the half-angle of the divergent part is also shown in Figure 8, while the dependence of average axial velocity on the restrictor outlet on the change of half-angle is shown in Figure 9.

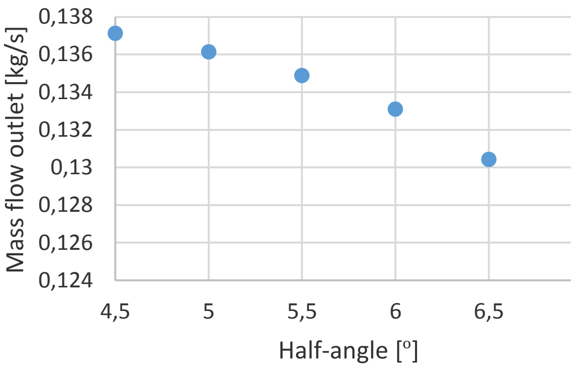

Fig. 8 *Dependence of mass flow on the outlet from half-angle*

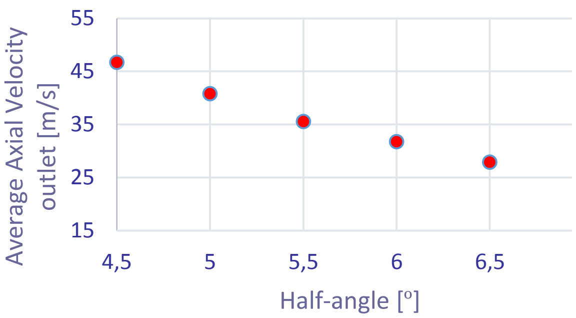

Fig. 9 *Dependence of average axial velocity on the outlet on the half angle*

As can be seen from the previous two pictures and table 1, the half angle affects the mass flow outlet and average axial velocity. With the increase of the half-angle, the outlet restrictor also increases, so that the mass flow rate and average axial velocity at the outlet also decrease, because these two quantities are calculated via the surface integral. As the half-angle value increases, the area of the restrictor output also increases. Out of the 6 analyzed cases, the best flow characteristic is achieved by the restrictor with the smallest half-angle, because in this case, the area of the restrictor outlet is the smallest. As a result of the CFD simulation, Figures 10 to 15 show contours of pressure for all analyzed cases, while Figures 16 to 21 show velocity streamlines for all considered cases.

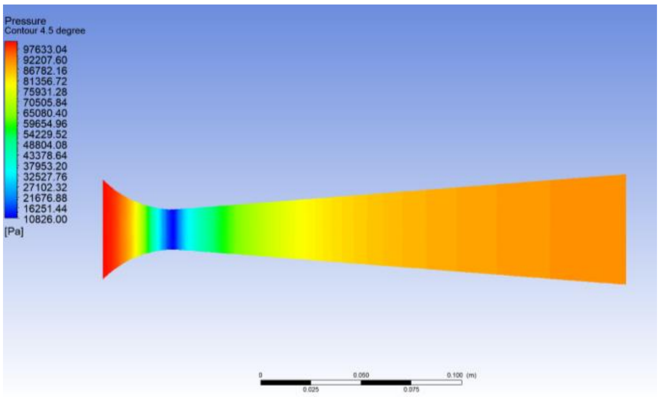

Fig. 10 *Pressure contour for half-angle of divergent part 4.5°*

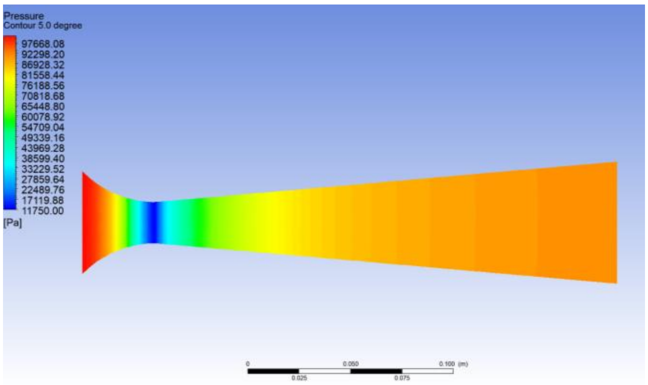

Fig. 11 *Pressure contour for half-angle of divergent part 5.0°*

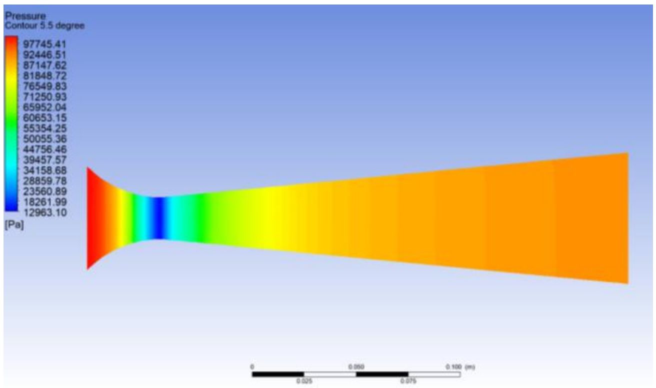

Fig. 12 *Pressure contour for half-angle of divergent part 5.5°*

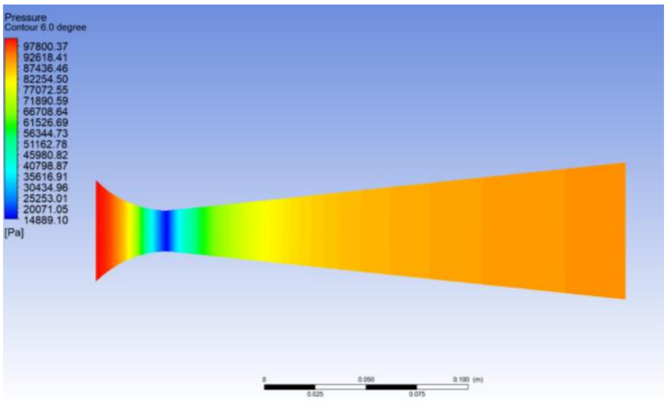

Fig. 13 *Pressure contour for half-angle of divergent part 6.0°*

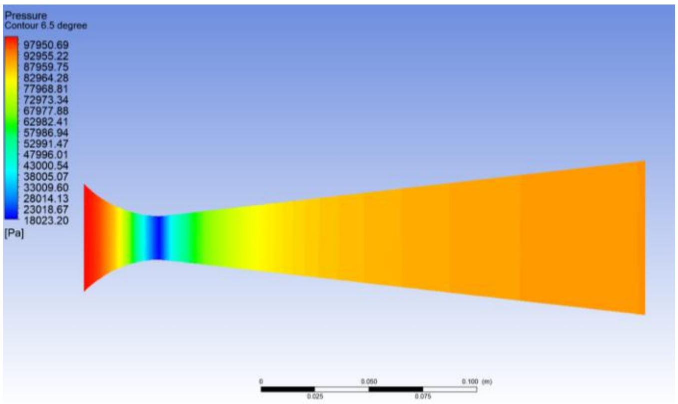

Fig. 14 *Pressure contour for half-angle of divergent part 6.5°*

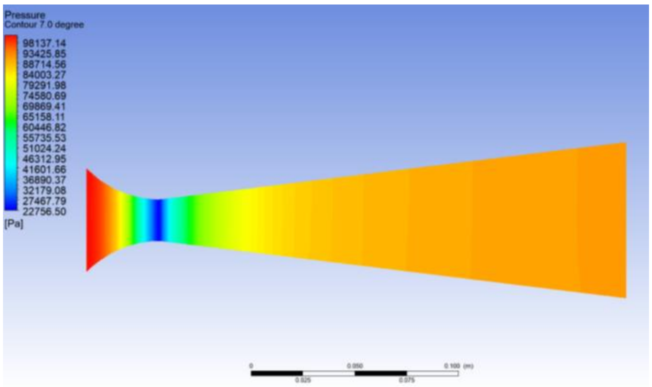

Fig. 15 *Pressure contour for half-angle of divergent part 7.0°*

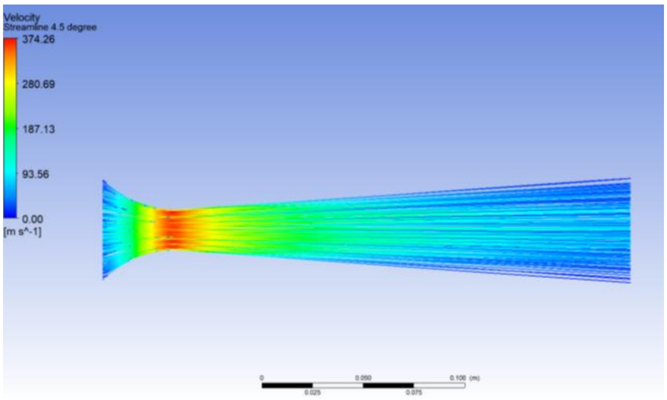

Fig. 16 *Velocity streamline for half-angle of divergent part 4.5°*

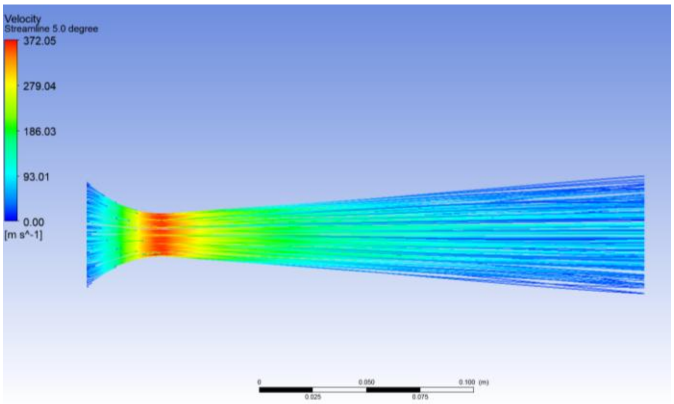

Fig. 17 *Velocity streamline for half-angle of divergent part 5.0°*

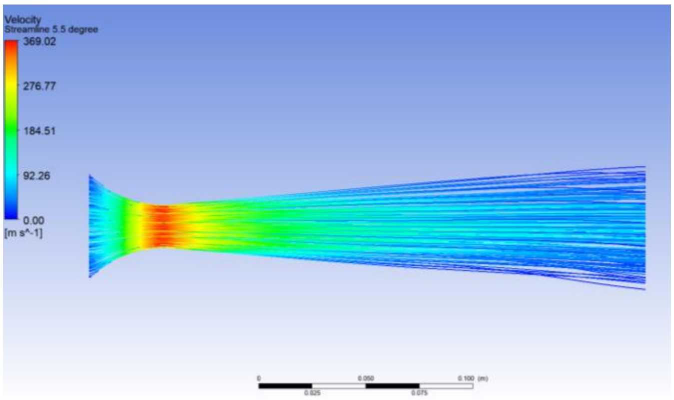

Fig. 18 *Velocity streamline for half-angle of divergent part 5.5°*

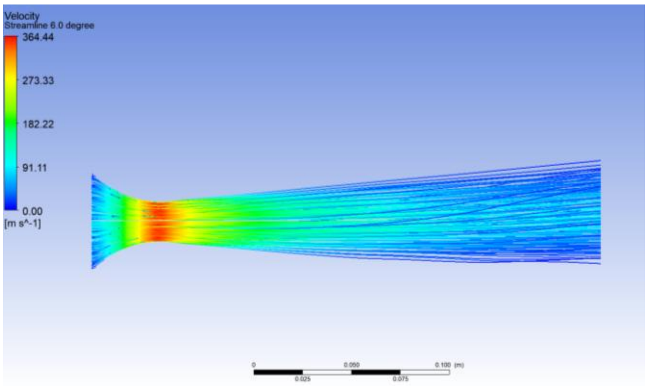

Fig. 19 *Velocity streamline for half-angle of divergent part 6.0°*

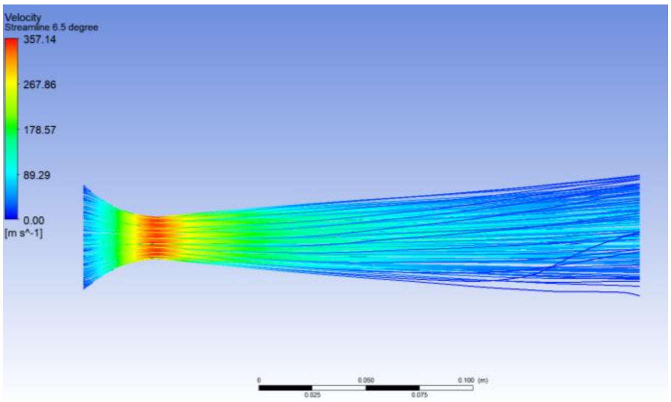

Fig. 20 *Velocity streamline for half-angle of divergent part 6.5°*

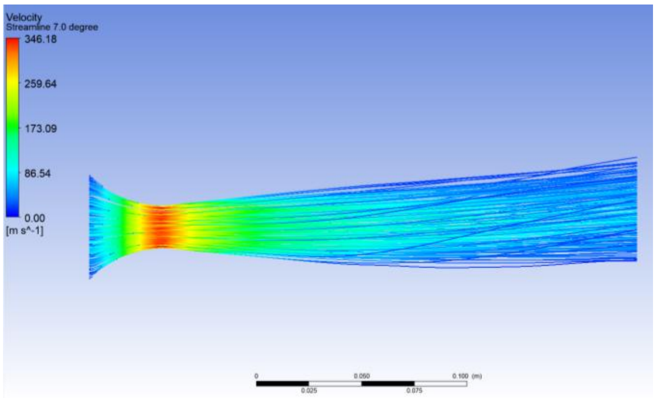

Fig. 21 *Velocity streamline for half-angle of divergent part 7.0°*

5. Conclusion

In this paper, a CFD simulation of airflow through the restrictor, which is a standard part of the air intake system of Formula Student vehicles, was performed. Through slow simulation, they tried to simulate airflow conditions as realistically as possible. The work aimed to examine the influence of the half-angle of the restrictor on the flow characteristic. With the increase of the half-angle, the area of the outlet from the restrictor increases, so it was confirmed through CFD simulation that with the increase of the half-angle, the mass flow at the outlet decreases, but also the average axial velocity at the outlet. It was also confirmed that CFD simulation can be used to perform very good analyses, which will be crucial when choosing the final design of the restrictor. If one had to choose one of the six analyzed half-angles, the design with a half-angle of 4.5° would be chosen. In general, it can still be concluded that a half-angle of 4.5°

will provide the best flow characteristic, but only from the mentioned six analyzed cases. Great attention should be paid to the design of the restrictor to ensure that the design restrictor with the best possible flow characteristics. What should be taken into account is that when designing the air intake system, attention should also be focused on the design of the exhaust system because only one well-designed system of these two cannot provide the maximum that can be extracted from the engine in terms of higher power and higher torque moment.

6. References

1. Versteeg, H. K., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: the finite volume method. Pearson education.
2. Mohamad, B., Karoly, J., & Zelentsov, A. (2020). CFD modeling of formula student car intake system. Facta Universitatis, Series: Mechanical Engineering, 18(1), 153-163.
3. Patidar, L., & Bhamidipati, S. R. (2014). Parametric study of drag force on a formula student electric race car using CFD. In Applied Mechanics and Materials (Vol. 575, pp. 300-305). Trans Tech Publications Ltd.
4. Christoffersen, L., Söderblom, D., & Löfdahl, L. (2009, July). Optimizing the cooling air flow of a formula car using cfd. In Proceedings of the European Automotive Simulation Conference.
5. Vivek, S., Pathak, R., & Singh, R. (2020). CFD Analysis, Analytical Solution, and Experimental Verification for Design and Analysis of Air Intake of Formula Student Car. In *Recent Trends in Mechanical Engineering: Select Proceedings of ICIME 2019* (pp. 553-566). Springer Singapore.
6. Kishore, R., Nivethan, M. H., Khumar, S. P., & Bharathi, A. A. (2022, November). Design and CFD analysis of undertray diffuser for formula student car. In AIP Conference Proceedings (Vol. 2446, No. 1, p. 180053). AIP Publishing LLC.
7. Dharmawan, M. A., Tjahjana, D. D. P., Kristiawan, B., & Pertiwi, S. I. (2020, April). Design and aerodynamics analysis of rear wing formula student car using 3 dimension CFD (computational fluid dynamics). In AIP Conference Proceedings (Vol. 2217, No. 1, p. 030166). AIP Publishing LLC.
8. Jackson, F. F. (2018). Aerodynamic optimisation of Formula student vehicle using computational fluid dynamics. Fields: journal of Huddersfield student research, 4(1), 40-65.
9. Wengrzyn, O. (2021). Validation of CFD predictions for flow over a full-scale formula student vehicle using PIV in real conditions. Journal of TransLogistics, 7(1).
10. Sedlacek, F., & Skovajsa, M. (2016). Optimization of an intake system using CFD numerical simulation. Proceedings in Manufacturing Systems, 11(2), 71.
11. Kamath, S. R., MP, P. K., Shashank, S. N., Damodaran, V., Anand, S. R., & Kulkarni, P. (2013). CFD and experimental optimization of formula SAE race car cooling air duct (No. 2013-01-0799). SAE Technical Paper.
12. Kang, N., & Yang, Y. (2014). Simulation and Analysis of Formula Racing Car's Diffuser Based on CFD Technology. In Applied Mechanics and Materials (Vol. 685, pp. 191-194). Trans Tech Publications Ltd.
13. Jadhav, J., Lad, S., Bhagat, P., More, K., & Patil, K. (2021). Study of Drag Around the Nose Cone of FSAE Vehicle using CFD Analysis. International Journal of Mechanical Handling and Automation, 7(1), 29-44.
14. Isiadinso, C. C. (2016). NUMERICAL SIMULATION AND ANALYSIS OF HEAT TRANSFER AND FLUID FLOW IN FORMULA STUDENT VEHICLE COOLING SYSTEM.
15. Permana, A. G. (2022). DESAIN DAN ANALISA AERODINAMIKA FRONT WING PADA MOBIL FORMULA STUDENT DENGAN MENGGUNAKAN SIMULASI CFD (COMPUTATIONAL FLUID DYNAMICS).
16. Jackson, F. F., & Stetsyuk, V. Aerodynamic Optimisation of Formula SAE Vehicle using Computational Fluid Dynamics.
17. Mohanan, A., Pandey, J. A., Gupta, A. V., Almeida, S. G., & Choudhari, D. Development of External Aerodynamics of an FSAE Racecar using Computational Fluid Dynamics.
18. Formula Student Rules. (2022). SAE
19. Lučić, M. (2022). Kinematic analysis of the slider-crank mechanism of an internal combustion (IC) engine using modern software. Mechanization in agriculture & Conserving of the resources, 68(1), 11-17.