IOP Conference Series: Materials Science and Engineering

PAPER • **OPEN ACCESS**

Aerodynamics Analysis of Mobil Irit Tarumanagara using CFD Method

To cite this article: Poppy Kesuma *et al* 2020 *IOP Conf. Ser.: Mater. Sci. Eng.* **1007** 012032

View the article online for updates and enhancements.

This content was downloaded from IP address 125.160.226.126 on 30/12/2020 at 09:56

Aerodynamics Analysis of Mobil Irit Tarumanagara using CFD Method

Poppy Kesuma*, Steven Darmawan, Agus Halim
Mechanical Engineering Department, Faculty of Engineering
Universitas Tarumanagara

* poppy.515160048@stu.untar.ac.id

Abstract. Aerodynamics on vehicles plays an important role in increasing the efficient vehicles. Nowadays, research in the field of aerodynamics of four-wheeled vehicles is carried out with the concept of optimization of the geometry shape of the vehicle. Some studies found that the best form in the aerodynamic aspect is the streamlined form, because it has the lowest resistance coefficient value, which is 0.04. Previously, the body of Mobil Irit Tarumanagara designed conventionally in Eco Vehicle concept, using experiments method. The experiments method is very complicated and inefficient, because engineers must draw the designs first, mock-up making, making negative molds, and then mold-casting. Aside from the processes, the aerodynamics performance of Mobil Irit Tarumanagara need to be analyzed. Therefore, this study will analyze the aerodynamics performance of Mobil Irit Tarumanagara using CFD (Computational Fluid Dynamic). CFD simulation will be done on 3D on 3 different body models and 5 speed variations: 1.39 m/s; 2.78 m/s; 4.17 m/s; 5.56 m/s; and 12.5 m/s. The mesh used is in tetrahedral form with a total of 76752 nodes in body no. 1; 55439 nodes in body no. 2; 42551 nodes in body no. 3. The results of the simulation presented by two flow parameters; velocity and pressure which shows the best body model is body 3, because it has the most aerodynamics performance, especially at the front and back of the body. The air velocity at front reaches 9.387 m/s; and vortices at the back flows in 3.271 m/s. This results serve as a reference for development on Mobil Irit Tarumanagara's body model

1. Introduction

Aerodynamics on vehicles plays an important role in increasing the efficiency of energy use of private vehicles. In this regard, research in the field of aerodynamics of four-wheeled vehicles is carried out with the concept of optimization of the shape of the vehicle. Some studies explain that the best form in the aerodynamic aspect is the streamlined form, because it has the lowest resistance coefficient value, which is 0.04.[1] Technology advancement can help engineers in vehicle design. The use of experiments as a method is very complicated and inefficient, because engineers must draw designs first, mock-up making, making negative mold, and then mold casting. Therefore, in this study, using numerical or computational methods, namely with applications that use the concept of 2D flow and 3D flow, such as CFD (Computation Fluid Dynamic).

Previously, the body of *Mobil Irit Tarumanagara* was designed and simulated using the CFD technique, and produced a Coefficient of Drag of 0.117398, a Coefficient of Lift of -0.372, and a frontal area of 5.721e + 5 mm^2.[2] The results of this study serve as a reference for the design and simulation to be carried out. This simulation will be done on 3D on 3 different body models with minimum of 2050 mm long, 680 mm width, and 625 mm height, and on 5

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

speed variations to describe on 2 parameters, velocity and pressure on the body. The purpose of this research is to design the body of Mobil Irit Tarumanagara with good aerodynamic performance, which can reduce energy that the car needed to accelerate.

2. Research Methodology

2.1. Research Flowchart

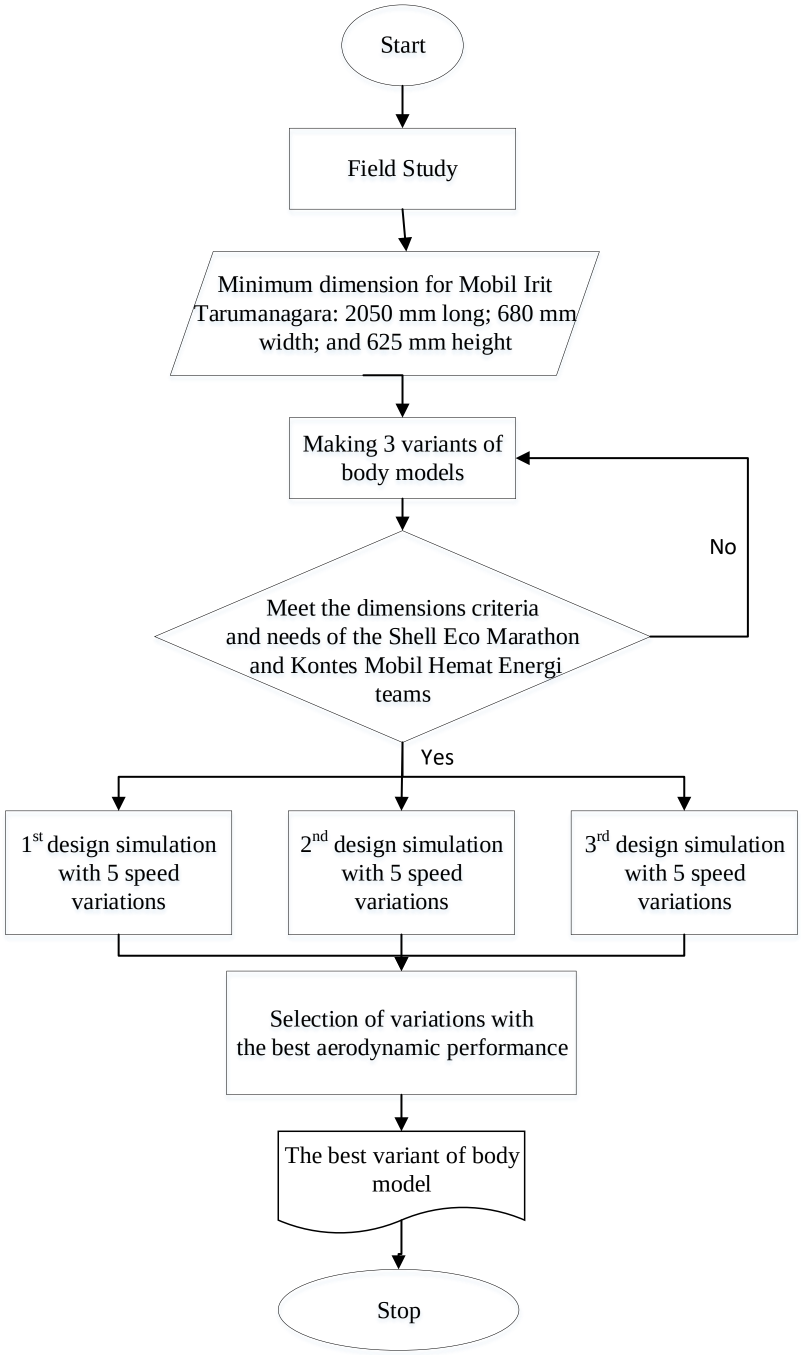

Figure 1. Research Flowchart

2.2. Boundary Conditions

Computational Domain with distance from inlet to the leading edge is 3 times the model length, trailing edge to the outlet is 5 times the model length, and computational domain height is 3 times height up the model, and 1 time height down the model.[3] The boundary condition in this simulation are adjusted to an external flow simulation. The boundary conditions are:

- Fluid Type : Air (Single Phase)
- Initial Pressure : 1 atm
- Initial Temperature : 300 K
- Air velocity on inlet with variations:

Table 1**.** Variations of air velocity on inlet

Variant	Velocity in m/s
1	1.39 m/s
2	2.78 m/s
3	4.17 m/s
4	5.56 m/s
5	12.5 m/s

2.3. Test Models

The following are 3 variations of the vehicle body design:

Figure 2. 1st body model in 3D

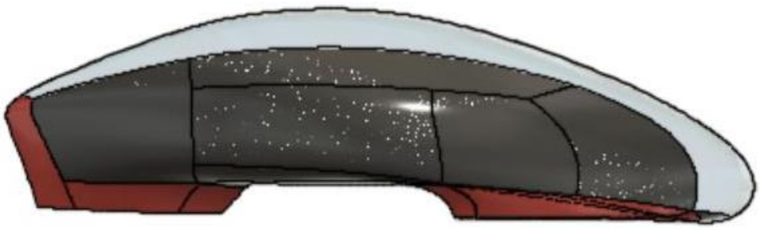

Figure 3. 2nd body model in 3D

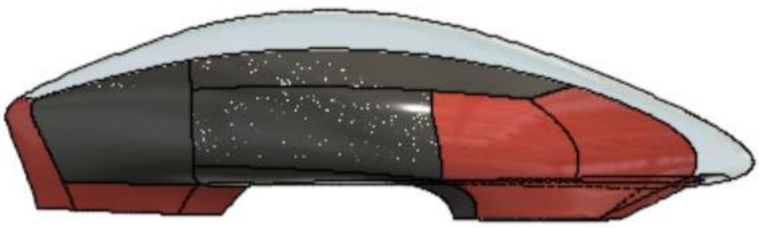

Figure 4. 3rd body model in 3D

Table 2. Technical specification of the models

Parameter	**1st Model**	**2nd Model**	**3rd Model**
Area	12.42E+06 mm^2	6.112E+06 mm^2	6.037E+06 mm^2
Volume	25.92E+08 mm^3	8.752E+08 mm^3	8.268E+08 mm^3
Length	2772 mm	906.097 mm	906.069 mm
Width	680 mm	818.727 mm	805.248 mm
Height	651.46 mm	2837.675 mm	2837.675 mm
Center of Mass	X : 0.103 mm, Y : 350.543 mm, Z : -385.735 mm	X : -2.21576 mm, Y : 361.732 mm, Z : -277.49 mm	X : -2.21583 mm, Y : 380.538 mm, Z : -232.022 mm

3. Results and discussion

The following are figures of the velocity simulation results for 1st, 2nd, and 3rd body models:

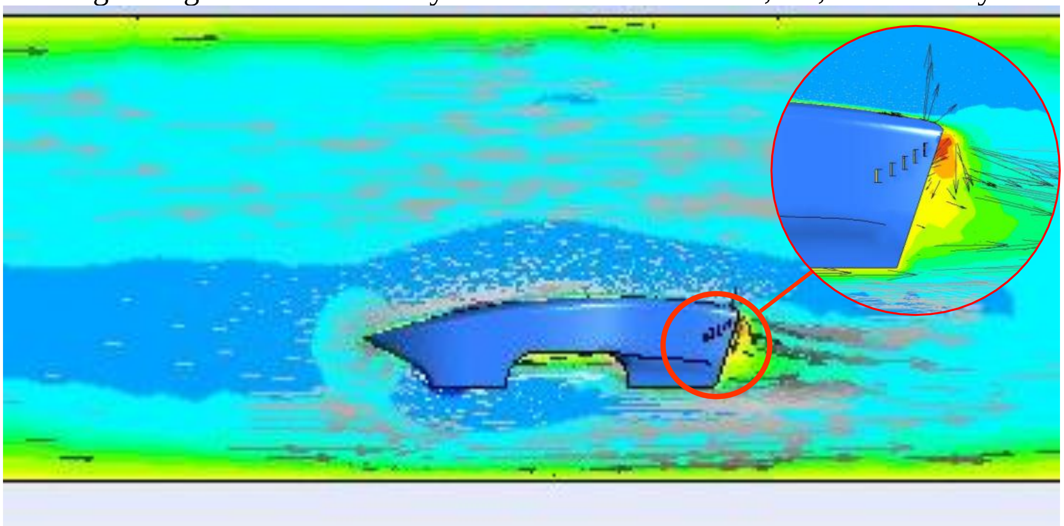

Figure 5. Velocity-w distribution of 1st body model

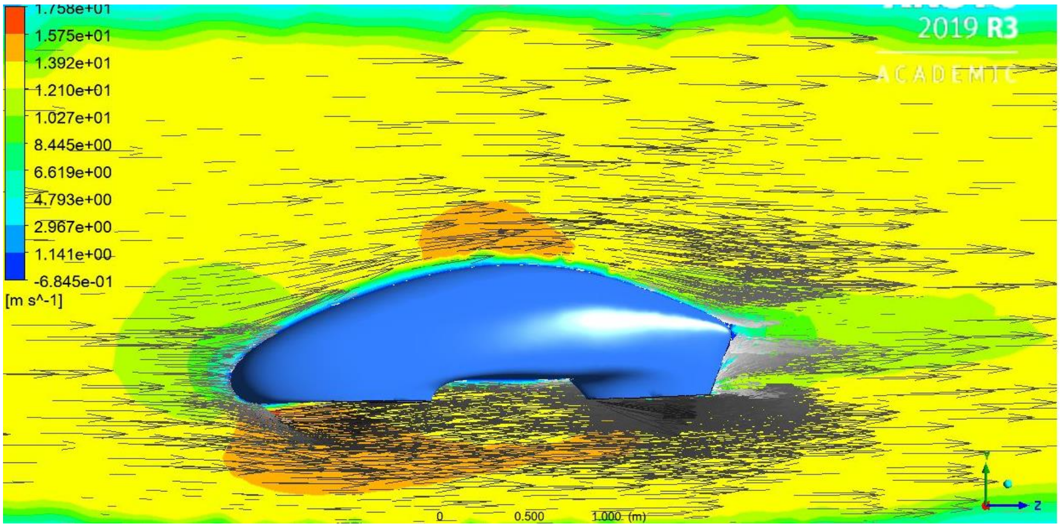

Figure 6. Velocity-w distribution of 2nd body model

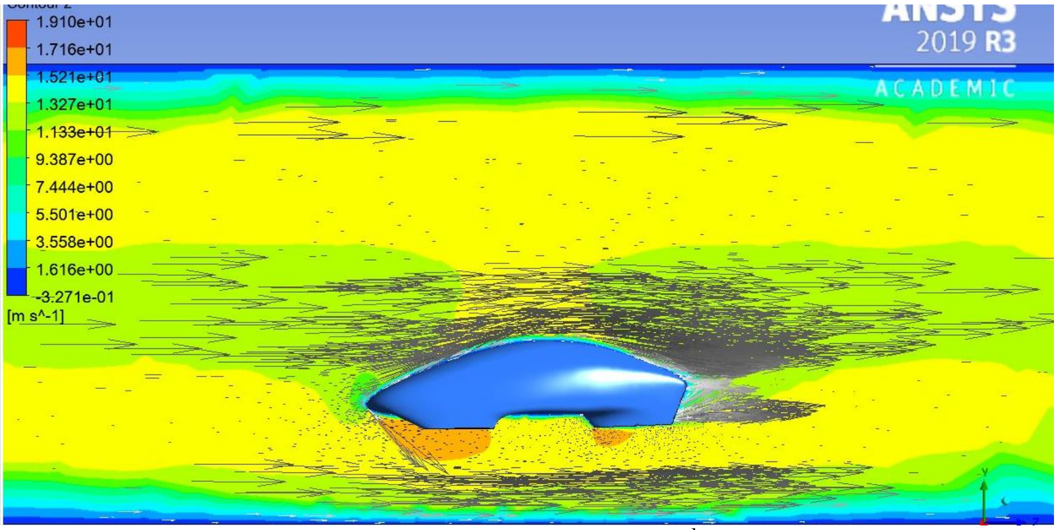

Figure 7. Velocity-w distribution of 3rd body model

Table 3. Velocity simulation results

No.	Inlet Velocity	Maximum Velocity			Minimum Velocity		
		Body 1	Body 2	Body 3	Body 1	Body 2	Body 3
1.	1.39 m/s	0.511 m/s	1.943 m/s	2.110 m/s	-2.018 m/s	-0.136 m/s	-0.079 m/s
2.	2.78 m/s	1.286 m/s	3.899 m/s	4.237 m/s	-4.055 m/s	-0.245 m/s	-0.012 m/s
3.	4.17 m/s	2.847 m/s	5.854 m/s	6.363 m/s	-8.134 m/s	-0.34 m/s	-0.053 m/s
4.	5.56 m/s	2.605 m/s	7.804 m/s	8.488 m/s	-8.132 m/s	-0.809 m/s	-0.035 m/s
5.	12.5 m/s	5.808 m/s	17.58 m/s	19.1 m/s	-18.28 m/s	-0.685 m/s	-0.327 m/s

As seen on the velocity-w on the 1st, 2nd, and 3rd models at Figure 5 and Figure 6, there are differences in the flow separation at the front and back area of the model. When a car moves forward, a high-pressure zone occurs because air flows in certain velocity and hit the front side of the car continuously. [4] And after the flow separates, the pressure decrease and a low pressure zone is created behind the car. This low pressure zone pulls the car from behind opposite to its moving direction and creates pressure drag. This low pressure zone is created due to the separation of flow and consequent vortices are generated at the back of the car.[5] Therefore, At the back of the model, wake occurs, and the velocity distribution occurs in this area decreasing, and causing an increase in pressure, thereby increasing the total resistance that occurs in the model. Wake is a low-pressure area due to a deficit of momentum due to separations. And generally wake can be found in the back area of the body.[6]

The following are figures of the pressure simulation results for 1st, 2nd, and 3rd body models:

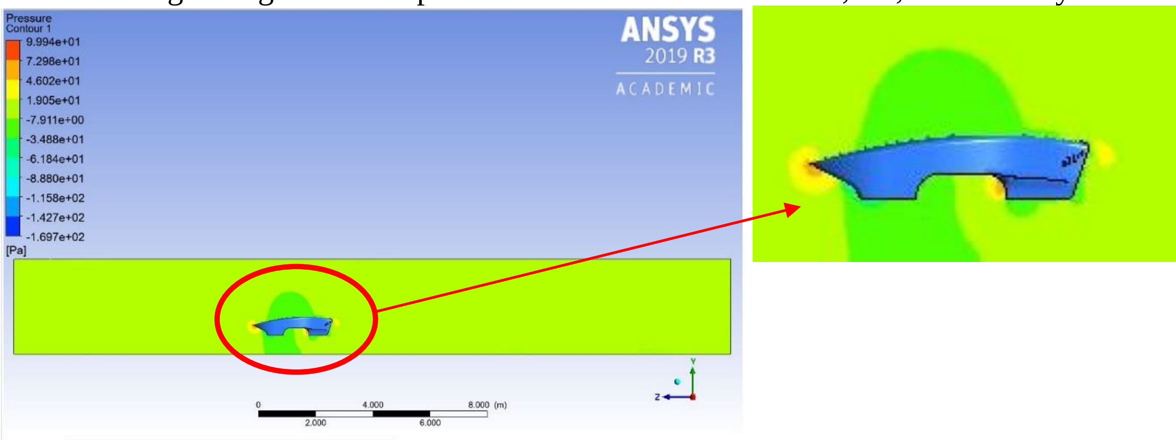

Figure 8. Result for pressure simulation of 3rd body model

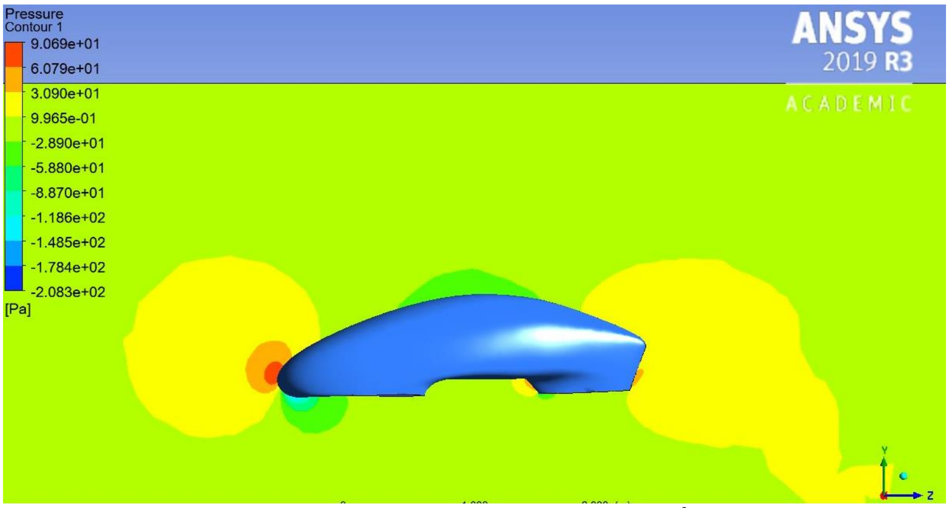

Figure 9. Result for pressure simulation of 3rd body model

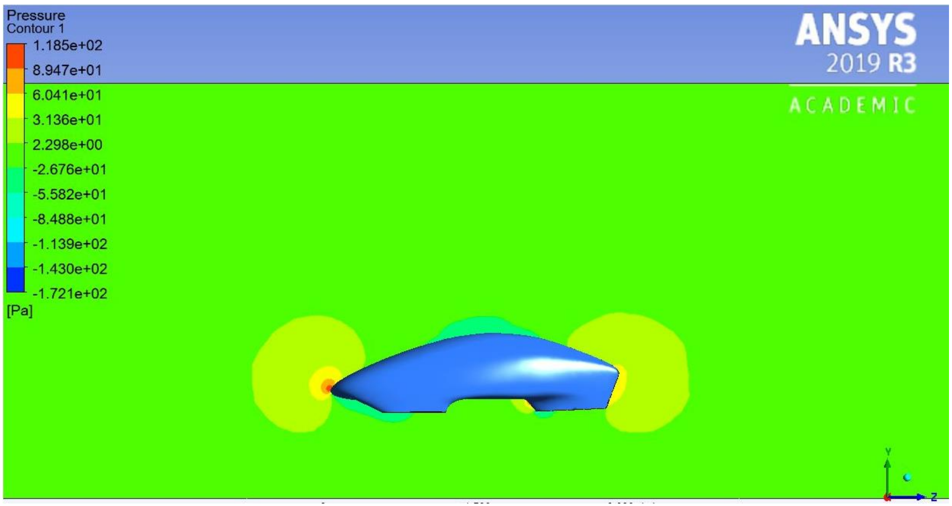

Figure 10. Result for pressure simulation of 3rd body model

Table 4. Pressure simulation results

No.	Inlet velocity	Maximum Pressure			Minimum Pressure		
		Body 1	Body 2	Body 3	Body 1	Body 2	Body 3
1.	1.39 m/s	1.242 Pa	1.129 Pa	1.469 Pa	-2.029 Pa	-2.499 Pa	-2.088 Pa
2.	2.78 m/s	4.957 Pa	4.497 Pa	5.880 Pa	-8.329 Pa	-10.17 Pa	-8.438 Pa
3.	4.17 m/s	1.979 Pa	10.11 Pa	13.22 Pa	-33.72 Pa	-23.04 Pa	-19.06 Pa
4.	5.56 m/s	1.979 Pa	17.95 Pa	23.49 Pa	-33.89 Pa	-41.09 Pa	-33.94 Pa
5.	12.5 m/s	9.994 Pa	90.69 Pa	118.5 Pa	-169.7 Pa	-208.3 Pa	-172.1 Pa

As seen on the results of pressure simulations on the 1st, 2nd, and 3rd body models, there are differences in pressure distribution in each model. As in simulations with a speed of 12.5 m/s, at the leading edge the pressure reaches 118.5 Pa; and at the top of the car, the pressure is at the minimum point, which is -172.1 Pa. Negative pressure means that there is a vacuum effect occurs to the body. The negative pressure created (negative only when compared to the atmosphere) sucks the car into the ground (downforce).[7] The air velocity at front is quite high as well, it reaches 9.387 m/s. And at the back, vortices flows in 3.271 m/s.

4. Conclusion

As seen on the velocity simulation results on the 1st, 2nd, and 3rd models, there are differences in the flow separation at the front and back area of the model. At the back of the model, wake occurs in results from the flow separation. Therefore, the speed distribution occurs in this area decreasing, and causing an increase in pressure, thereby increasing the total resistance that occurs in the model. The pressure simulation results can strengthen the analysis on the velocity simulation, because the result shows that the front area of the body and the back of the body is the highest pressure point, while the area above and under the body is the lowest pressure point. Thereby, in body 1 the maximum velocity is only at 5.808 m/s, lower than the inlet velocity, which is 12.5 m/s. unlike the speed at body 3, the maximum velocity reaches 19.1 m/s, higher than the inlet velocity. So it can be said, that the wind resistance occurs in body 3 is smaller, and body 3 has better aerodynamic performance.

5. References

[1] A. A. Hakim, "PENGARUH PENAMBAHAN ATAP SEKUNDER KABIN MOBIL (SECONDARY CABIN ROOF) TERHADAP GAYA AERODINAMIS DAN PERILAKU ARAH PADA MOBIL SEDAN," 1395.

[2] Universitas Tarumanagara, "Laporan Kendaraan Mobil Irit Tarumanagara," 2019.

[3] H. K, Versteeg dan W. Malalasekera, *Introduction: Computational Fluid Dynamics The Finite Volume Method*, vol. 44, no. 2. 2006.

[4] E. Guilmineau, "Computational study of flow around a simplified car body," *J. Wind Eng. Ind. Aerodyn.*, vol. 96, no. 6–7, hal. 1207–1217, Jun 2008, doi: 10.1016/J.JWEIA.2007.06.041.

[5] S. M. Rakibul Hassan, T. Islam, M. Ali, dan M. Q. Islam, "Numerical study on aerodynamic drag reduction of racing cars," *Procedia Eng.*, vol. 90, hal. 308–313, 2014, doi: 10.1016/j.proeng.2014.11.854.

[6] D. Variasi dan S. Diffuser, "Studi numerik karakteristik aliran bagian rear-end bus penumpang dengan variasi sudut diffuser," vol. 1, no. 1, hal. 1–5, 2012.

[7] K. E. Y. Information, "Formula 1 The Knowledge," 2016.