Aerodynamic Analysis of the Rear Spoiler of Hatchback Vehicles Using SST K-ω Turbulence Model

Andre Kuwito, Steven Darmawan, and Harto Tanujaya

Abstract In the development of the times and technology, demands for forms and characteristics have a good aerodynamic value. As a result, various types of new models are emerging rapidly in the automotive market. In addition, many drivers modify their cars. One of them is by adding a body kit or just adding a spoiler to make it more esthetic. As time goes by, many car manufacturers pay attention to several aspects in designing their cars. One of them is aerodynamic. The aerodynamics performance of a car is affected by the drag coefficient (Cd) or drag when the car is in motion. Furthermore, to get this Cd value, a study will be carried out that focuses on the effect of adding spoilers with NACA 2412 on vehicles. The research can be carried out using the CFD simulation. In the study, a simulation was carried out using a Hatchback car and two spoiler models with a NACA 2412 airfoil profile. The result obtained is that a car without a spoiler produces a Cd of 0.562, and for a car model with a spoiler 1, it produces a Cd of 0.383, while for a car with a spoiler of 2 it produces a Cd of 0.409.

Keywords Aerodynamics · Car · CFD · K-ω turbulence model · Rear spoiler

1 Introduction

Aerodynamics is defined as a change in the movement of an object due to air resistance when moving at high speeds. The aerodynamics of a car is also called the drag coefficient (Cd) or drag when the car is in motion. Aerodynamics is inseparable from

A. Kuwito (✉) · S. Darmawan · H. Tanujaya
Universitas Tarumanagara, Jakarta, Indonesia
e-mail: andre.515180030@stu.untar.ac.id

S. Darmawan
e-mail: stevend@ft.untar.ac.id

H. Tanujaya
e-mail: hartotan@ft.untar.ac.id

Irwansyah et al. (eds.), *Proceedings of the 4th International Conference on Experimental and Computational Mechanics in Engineering*, Lecture Notes in Mechanical Engineering, https://doi.org/10.1007/978-981-99-7495-5_17

the science regarding the profile of the Naca airfoil. Spoiler is a car component that affects the aerodynamics of a car, Rear spoiler is responsible to the drag and downforce experienced by the car. Spoilers work by applying downforce or downward pressure force on the car to reduces the lift generated by high speed [1].

In particular, rear spoiler of a car come in many geometries based on the function. One could come from the dealership, or as accessories. The application of rear spoilers on a car is also often related to the esthetics of the vehicle, especially in hatchback type. Application of inappropriate rear spoiler geometry may relate to safety. Rear spoiler effect on a car can be analyzed with experimental methods using wind tunnel, which usually complicated, limited only for some geometries and costly. Other way to analyze the effect of rear spoiler is by using CFD method [2]. As the next step of this Cd analysis research, a study will be conducted that focuses on the effect of adding spoilers with NACA 2412 on a hatchback type vehicle, the vehicle to be used is the third-generation VW Scirocco [3–5]. The choice of this vehicle is because the model of the vehicle is attractive and quite in demand in the market. The formulation of the problem in this study is how the value of Cd with the use of spoilers with the Naca 2412 airfoil profile [6–8]. The formulation of other problems is what factors affect Cd in vehicles and how to analyze the fluid flow rate in vehicles.

In this study, the limitation of the problem used was a car of the hatchback type and the speed of the vehicle used started from 60 km/h. To obtain the results of this study, it will be compared the Cd value produced between the car with the addition of spoilers with NACA 2412 and with those that are not. The purpose of this study is to analyze the influence of aerodynamics of vehicle performance by using rear spoilers and to determine the effect of using spoilers with NACA 2412 on the downforce of hatchback type vehicles [9].

1.1 Problem Formulation

Looking at the problems that arise, it can be known what to do, namely:

1. How does the use of spoilers with NACA 2412 airfoil profile affect the Cd value?
2. How is the analysis of fluid flow rate in vehicles?
3. What factors affect Cd in cars?

2 Research Method

Analysis of the study of Cd on steel plate with airfoil NACA 2412 profile was carried out by experimental method. Experiments were carried out to determine the aerodynamic performance of the plate and also the best mesh configuration. This research uses computational fluid dynamics (CFD) simulation. This simulation uses the ANSYS Student 2022 R1 and Autodesk Fusion 360 software which is carried out on an Acer Aspire E14 laptop. The simulation is divided into three stages, starting with the geometric model process, then the CFD model, and the last one is post-processing [10, 11].

2.1 Geometry Model

In this study, it used a hatchback type vehicle. An example car used is the third-generation VW Scirocco car. The VW Scirocco has dimensions of 4256 × 1810 × 1404 mm. The creation of this geometry is done using the Autodesk fusion 360 application (Figs. 1, 2).

Next is the spoiler design process which is also carried out on the Autodesk fusion 360. The spoiler is designed by utilizing the shape of the NACA 2412 airfoil profile, and the dimensions of the spoiler are 1800 × 150 mm as seen on Fig. 3.

Fig. 1 VW Scirocco [12]

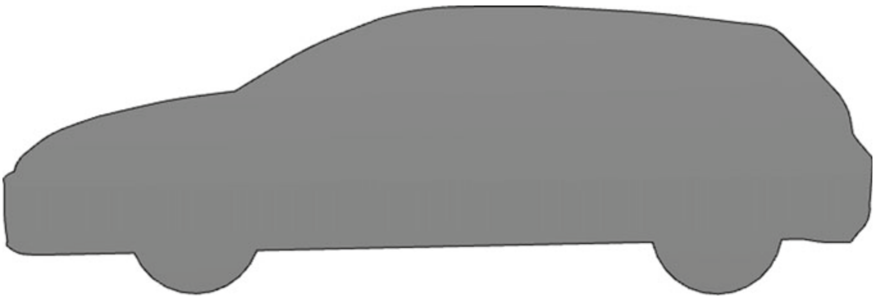

Fig. 2 Car geometry design

Fig. 3 Spoiler model

The computational domain used for simulation has a length of 1418.64 mm, a width of 784.73 mm, and a height of 463.16 mm with details of the length of the front wall, right side, left side, and top which is two times the size of the car length. For the rear length, it is 5 times the length of the car. And the bottom wall is 50 mm.

2.2 CFD Models and Configuration

CFD simulation is conducted by using Ansys Fluent Student Edition. In the simulation process of this vehicle, a 0.005 m mesh is used and with a type of mesh, namely tetrahedral SST *k-ω turbulence model is used* [13]. For the vehicle simulation process, the velocity used is 16.66 m/s or 60 km/h, the density of air is 1.225 kg/m^3, and the temperature is 288.16 K. After carrying out the feeding meshing process, the number of nodes was obtained by 87,382 and the number of cells was 469,049 (Fig. 4).

The car used for the simulation process will be reduced by a scale of 1:24 or reduced by 24 times its original size. This is intended to simplify the CFD simulation process that will be carried out. After the vehicle and spoiler design process is carried out, the process continued to assembling the two component. The spoiler is placed on the back of the car with a slope level of 25°, and for the second spoiler, it is placed 150 mm from the vehicle body. CFD simulation conducted three-dimensionally with single-phase air. In order to simplify the simulation process, the geometry model is

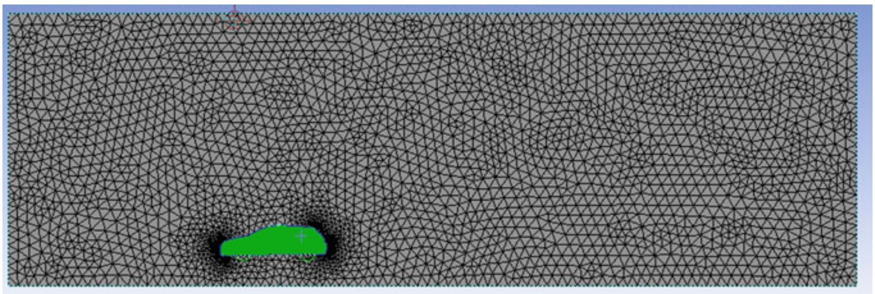

Fig. 4 Meshing results

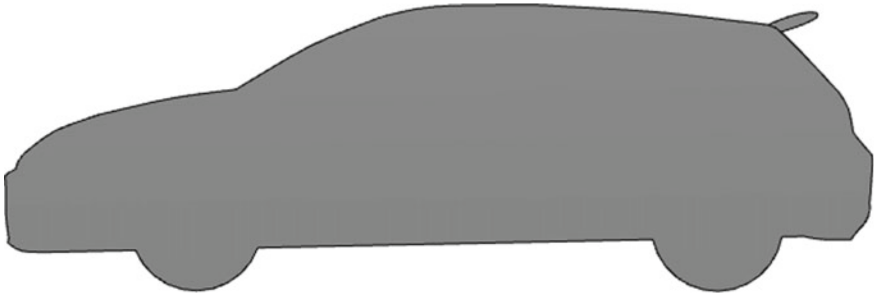

Fig. 5 Car configuration 1

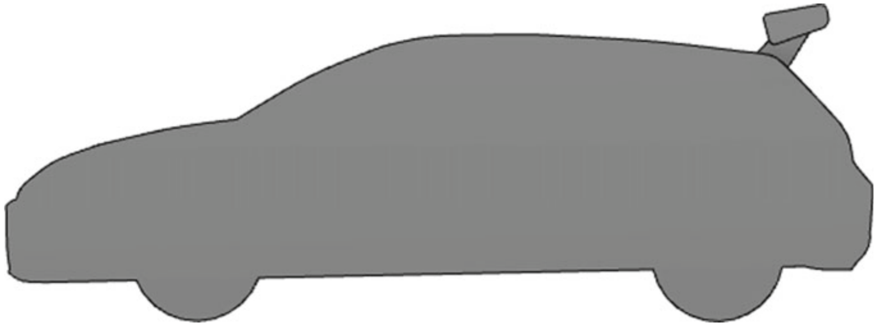

Fig. 6 Car configuration 2

reduced 24 times than its original size. Since the right and left side geometry of the model is identic, the simulation is assumed to be symmetric (Figs. 5, 6).

2.3 Turbulence Model

This k-omega turbulent SST model is a k-omega turbulent model that has been refined several times due to its weakness in sensitivity to free stream boundary conditions. In the model refined by Menter, the k-omega model is combined with the k-epsilon model, so that it has good capabilities in the area around the wall and a low Reynold number as an advantage of k-omega, and flows with adverse pressure gradients and insensitivity in the free stream area which is the advantage of k-epsilon.

$$\frac{\partial}{\partial_t}(\rho k) + \frac{\partial}{\partial_{x_i}}(\rho k u_i) = \frac{\partial}{\partial_{x_j}}\left[\tau_\omega \frac{\partial k}{\partial_{x_j}}\right] + G_k - y_k + S_k \tag{1}$$

$$\frac{\partial}{\partial_t}(\rho \underline{\omega}) + \frac{\partial}{\partial_{x_j}}(\rho \underline{\omega} u_j) = \frac{\partial}{\partial_{x_j}}\left[\tau_\omega \frac{\partial \varepsilon}{\partial_{x_j}}\right] + G_\omega - Y_\omega + D_\omega + S_\omega \tag{2}$$

3 Result

The final result of the simulation will later compare the best Cd values of cars without spoilers, cars with spoilers 1, and cars with spoilers 2. The simulation will be carried out by providing a fluid flow inlet on the X axis.

3.1 CFD Simulation Results on Static Pressure Contours

In the simulation results carried out on the three car models, it can be seen that Fig. 8 which is the result of static pressure simulation for cars without spoilers has a fairly small static pressure with a static pressure value of 1616 Pa compared to two car models with other spoilers. Figure 9 illustrates the greatest spread of static pressure on the front bumper, windshield, and in the spoiler section of the car with a value of 1942 Pa. Meanwhile, Fig. 10 which depicts a simulation of a car with spoiler 2 shows the largest spread of static pressure points in the front bumper, windshield, and legs that connect the spoiler and car body with a static pressure value of 1720 Pa.

By looking at the contours of static pressure in Figs. 7, 8, and 9, it can be seen that with the addition of a spoiler, it can affect the static pressure on the car. The greater the static pressure, the greater the downforce on the car. This happens because static pressure occurs in the spoiler and the direction of the static pressure is toward the bottom. So the greater the static pressure that exists, the greater the downforce value, with an average value increase of 0.63%.

3.2 CFD Simulation Results on Velocity Contours

In the simulation with a car model without a spoiler depicted in Fig. 11, the speed produced after the separation process is equal to the inlet speed of 16.66 m/s. This event also affects the resulting wake being less massive. This happens because the data collection point chosen is the point that will be affected by the presence of a spoiler, so that in a car simulation without a spoiler, the resulting speed does not have an impact on body separation.

In the second simulation, namely a car with spoiler 1, it can be seen in Fig. 12, the speed produced after the flow separation process is 2.55 m/s. The speed is reduced due to the presence of a large Wake, so that it has an impact on reducing the flow speed.

In the third simulation using a car model with spoiler 2 whose results can be seen in Fig. 13, the resulting speed is 12.85 m/s. This happens because the Wake produced after separation of the flow is not too large, so that it has an impact on the speed on the back of the car.

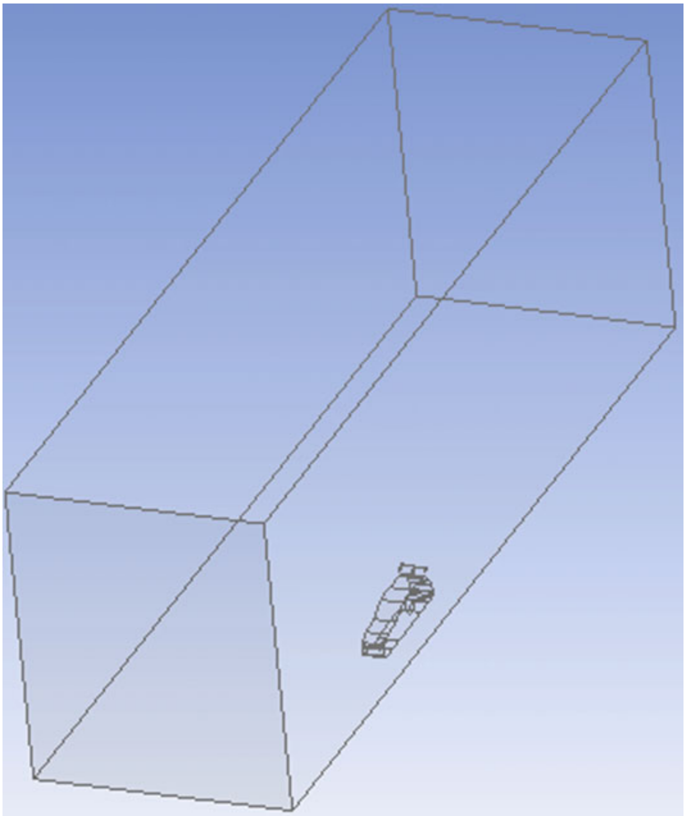

Fig. 7 Symmetry domain

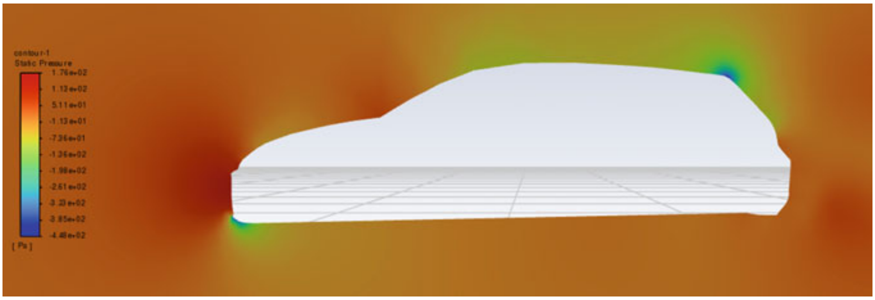

Fig. 8 Static pressure car without spoiler

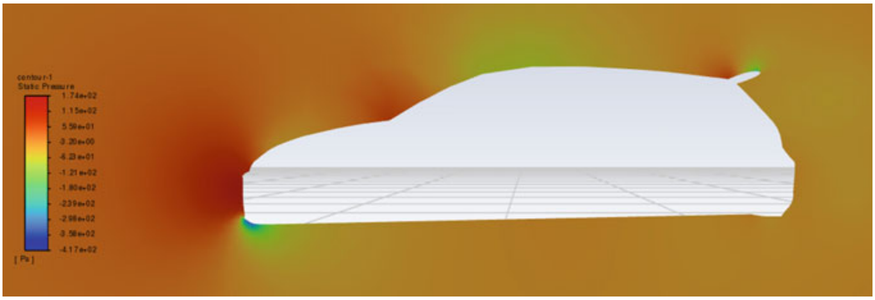

Fig. 9 Static pressure generated by spoiler model 1

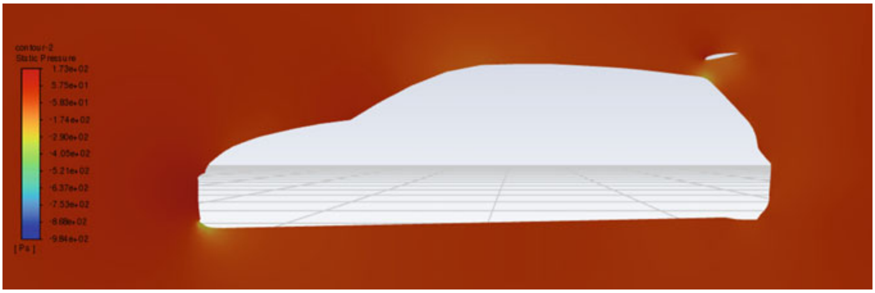

Fig. 10 Static pressure generated by spoiler model 2

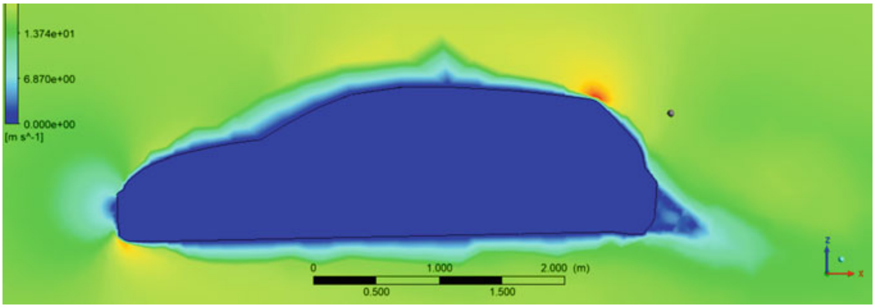

Fig. 11 Car velocity contours without spoiler

3.3 CFD Simulation Results on Aerodynamic Force

Based on the presentation of simulation data and calculations that have been carried out above, it can be seen that the Cd value generated by cars with spoiler 1 has a smaller value of 0.383 while cars without spoilers have a Cd value of 0.562; this indicates that the Cd on cars with spoiler 1 depreciated by 31.73%, while in cars with spoiler 2 which have a Cd value of 0.409, it depreciated from cars without spoilers

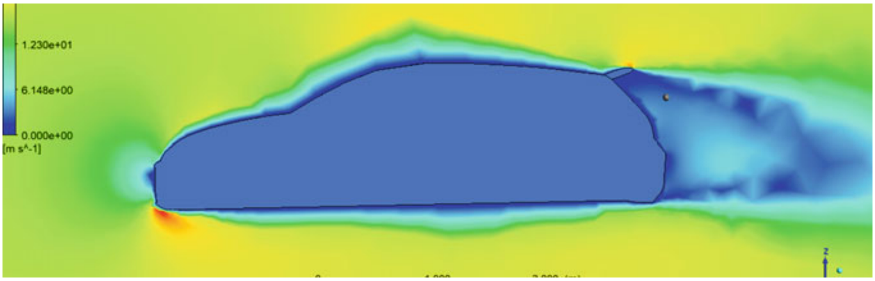

Fig. 12 Car velocity contour with spoiler 1

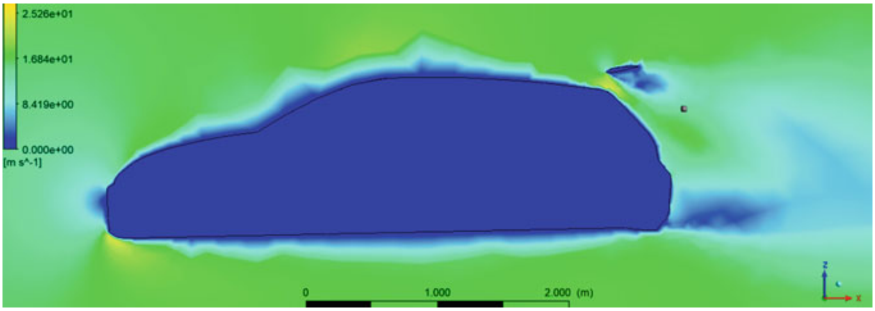

Fig. 13 Car velocity contour with spoiler 2

as much as 27.13%. This CFD simulation results related on Cd and Cl provided in Table 1.

From this, it can be said that the addition of spoilers has proven to be able to reduce the value of Cd which makes drag on vehicles improve. This is also evidenced by the European Journal of Computational Mechanics with its journal entitled "Reduce drag Coefficient of a Hatchback car utilizing fractional factorial design algorithm", suggesting that cars with spoilers can produce Cd that are less valuable than cars that do not have a spoiler [14]. The journal simulates a car with a hatchback type and produces a Cd on a 0.282-sized car and a 0.411 non-spoiler car.

In addition to Cd, there are simulation results in the form of lift or cl coefficients, the value of this coefficient will decrease as static pressure increases. This can be proven by the cl value in cars with spoilers that have been reduced by 5.47% compared

Table 1 Drag coefficient and lift coefficient data

No	Variations	Cd	Decrease value of Cd	Cl	Decrease value of Cl
1	Without spoiler	0.562	–	0.737	–
2	Spoiler 1	0.383	31.73%	0.696	5.47%
3	Spoiler 2	0.409	27.13%	0.698	5.21%

to cars without spoilers with a static pressure value of 1942 Pa. As for the cl value on cars with spoiler 2, it was reduced by 5.21% with a static pressure value of 1.720.

4 Conclusion

Based on the simulation results with the CFD method and the presentation of data that has been previously provided, it can be concluded:

1. Based on the simulation results on three car models with a speed of 16.66 m/s or 60 km/h, it can be concluded that adding a spoiler can make drag on the car improve, and this can be seen in Fig. 11 describing the results of the car velocity simulation with spoiler 1 in addition to that in Fig. 8 explaining that the vehicle with spoiler 1 has the most static pressure among 2 other car models, namely 1.942 Pa which will make the compressive force down the better.
2. Wake created by the separation of the flow of the vehicle body can cause a drag effect on the vehicle the greater and longer the Wake created, the better the drag on the vehicle and vice versa the smaller and shorter the Wake created by the separation of the vehicle body flow, the worse the drag that exists on the vehicle.
3. Taking into account the aerodynamic forces in vehicles, the NACA 2412 airfoil chosen as the basic form of spoiler manufacturing was shown to increase drag on the vehicle. This can be viewed from the Cd value generated by each vehicle model. Cars without spoilers have a Cd value of 0.562, for cars with spoiler 1 have a Cd value of 0.383, and for cars with spoiler 2 have a Cd value of 0.409. This shows that the use of spoilers with a NACA 2412 airfoil profile can reduce the Cd produced by cars without spoilers.

References

1. Cakir M (2012) CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle. St Cl Univ Sch Commons pp 1–72. Retrieved from http://scholarcommons.scu.edu/mech_mstr
2. Kesuma P, Darmawan S, Halim A (2020) Aerodynamics analysis of mobil irit tarumanagara using CFD method. IOP Conf Ser Mater Sci Eng 1007(1). https://doi.org/10.1088/1757-899X/1007/1/012032
3. Eken S (2019) Free vibration analysis of composite aircraft wings modeled as thin-walled beams with NACA airfoil sections. Thin-Walled Struct 139:362–371. https://doi.org/10.1016/j.tws.2019.01.042
4. Yossri W, Ben Ayed S, Abdelkefi A (2021) Airfoil type and blade size effects on the aerodynamic performance of small-scale wind turbines: Computational fluid dynamics investigation. Energy 229. https://doi.org/10.1016/j.energy.2021.120739
5. Palanivendhan M, Chandradass J, Bannaravuri PK, Philip J, Shubham K (2021) Aerodynamic simulation of optimized vortex generators and rear spoiler for performance vehicles. Mater Today Proc 45:7228–7238. https://doi.org/10.1016/j.matpr.2021.02.537

6. Dey S, Saha R (2020) Performance of NACA cambered and symmetrical airfoils as rear spoilers on race car aerodynamic drag & lift: computational fluid dynamics study. Proc Int Exch Innov Conf Eng Sci 6:292–298. https://doi.org/10.5109/4102505
7. Chodagudi KBR (2012) Analysis Of Naca 2412 for automobile rear spoiler using composite material certain applications. Int J Eng Res Technol 1(7):1–5
8. No R (1931) The national advisory committee for aeronautics. Science 74(1923):451. https://doi.org/10.1126/science.74.1923.451
9. Dey S, Saha R (2018) CFD Study on aerodynamic effects of NACA 2412 air foil as rear wing on a sports car CFD study on aerodynamic effects of NACA 2412 airfoil as rear wing on a sports car. no. December, pp 0–6. Retrieved from https://www.researchgate.net/project/CFD-Study-on-Aerodynamic-Effect-of-NACA-2412-Airfoil-as-Rear-Wing-on-a-Sports-Car
10. Taha TR (2005) An introduction to parallel computational fluid dynamics. 6(4). https://doi.org/10.1109/mcc.1998.736434
11. Fluent A, "Ansys Fluent Theory Guide," *ANSYS Inc., USA*, vol. 15317, no. November, pp. 724–746, 2013.
12. "Volkswagen Scirocco GTS teased." https://www.motortrend.com/news/1504-volkswagen-scirocco-gts-teased/ (accessed Jul. 17, 2022).
13. K.-J. Bathe, *Computational fluid and solid mechanics 2003[Recurso electrónico] :] proceedings, Second MIT Conference on Computational Fluid and Solid Mechanics, June 17–20, 2003*. 2003. [Online]. Available: http://0-www.sciencedirect.com.jabega.uma.es/science/book/9780080440460
14. Vahdati M, Beigmoradi S, Batooei A (2018) Minimising drag coefficient of a hatchback car utilising fractional factorial design algorithm. Eur J Comput Mech 27(4):322–341. https://doi.org/10.1080/17797179.2018.1550962