Numerical simulation of surface curvature effect on aerodynamic performance of different types of airfoils

Adel Muhsin Elewe
Project and construction department, Mustansiriyah University, Iraq
adelmuhsin@uomustansiriyah.edu.iq

Abstract:
The development of car modification using rear wings or spoilers, underlies the research on the aerodynamic performance of airfoil. The influence of aerodynamic forces will produce a down force to the bottom of the vehicle. The present paper investigate airfoil NACA 0012, NACA 4415, GOE 528 and GOE 652 to estimate the effectiveness of airfoil shape based on different angles of attack. The numerical simulation has been done using ANSYS Fluent to obtain drag coefficient, lift coefficient and lift-to-drag ratio of all cases. In addition, the drag and down forces have been calculated with respect to airfoil geometry features. The main objective in this paper is finding the effect of airfoil surface specification as a part of airfoil geometry features. The results observe that the highest lift coefficient value was achieved by GOE 652 which was equal to CL=1.9310 at 7 degree angle while the highest lift-to-drag ratio achieved by NACA 4415 which was equal to 100.8359. The highest down force was 1906.847 N at 15 degree by GOE 652 airfoil. The results show that the airfoil GOE 652 has the most effective surface area among the four airfoils.

Keywords: aerodynamic forces, airfoil, geometry surface features

1. Introduction

Aerodynamic forces represents the main factor that optimize vehicle fuel consumption and performance. The negative lift or down force and drag force considered the essential concepts in aerodynamic forces of vehicle design. The geometry of the vehicle body considered as one of the main factors for vehicle performance optimization due to the variation in forces caused by shape elements and air flow characteristics. The effect of these forces depends upon the size and shape of the objects in addition to the flow impact on it [1]. Spoilers and wings represent an essential added devices which capable to govern the air flow around the vehicle. The function of the spoiler is to reduce the amount of turbulence around the vehicle. Thus, it helps to reduce air drag for that, it is fitted to high performance sports cars [2] [3]. The significance of shape parameterization is expanded to the combination of part structure design with
aerodynamic specification design. The presses of air force generated by spoiler or wing on the road can be developed by arising car speed. Developing optimal airflow needs to investigate airfoil shapes with respect to its work mechanism [4] [5]. For that, the airfoil concept developed to represent the cross section of a body that is inserted in an airstream produces an aerodynamic force [6]. Analyzing the fluid flow around the airfoil enables to investigate the aerodynamic characteristics of airfoil. These characteristics of flow are typically calculated using Reynolds Averaged Navier–Stokes equations (RANS). The RANS approach presents an acceptable solution of calculations accuracy and computational resources [7]. The main geometry features of airfoil are the chord, camber and thickness. These features affect the performance characteristics of airfoils due to the changes in lift, drag, pressure distribution and moment of aerodynamic changes [8]. In this paper, CFD analysis of different airfoil models are presented by ANSYS software to investigate the airfoil performance characteristics. The goal of this study is testing different types of airfoil orientation to develop a specific data for finding optimal airfoil for spoiler or wing design that can balance the down and drag forces under various conditions of speed. The work developed by testing for types of airfoils to find the drag force, down force, drag coefficient and lift coefficient which represents the main factors of the aerodynamic characteristics. Then correlate the testing results with the airfoil geometric features to find the optimal airfoil area that can be used to optimize the cars performance.

2. Aerodynamic Forces

The aerodynamic specification is a process of achieving the best compromise between high inverse lift or down force and low drag force. It is dynamics related with flow of air. The aerodynamic effects classified into two type external and internal aerodynamics. Internal aerodynamics concern with flow of air within ducts and external aerodynamics concern with flow of air over a body [9]. The down force is the perpendicular air effect to the direction of flow. While the drag is the force generated the parallel effect of air flow. Evaluating the airfoil based on various angles, the lift and drag will expressed by dimensionless quantities. In order to model the airfoil, the coefficients (force and drag) must be specified. They represent the complex dependencies of inclination, shape and flow conditions. They are a measure of how good the aerodynamic shape based on lower drag or a lower air drag coefficient. The drag performance is characterized by the drag coefficient (CD). It is equal to the drag force divided by the environmental effect of the airfoil area [3]:
where, D is the Drag force, V is the flow speed, \( \rho \) is the density of fluid, and S is the reference area. This non-dimensional value allows to compare the drag performance between different setups and different vehicles. The lift of the vehicle is also characterized by the lift coefficient (CL). The lift coefficient is equal to the lift force divided by the environmental effect of the airfoil area and is defined as [8].

\[
C_L = \frac{L}{(1/2)\rho V^2 A}
\]

where, L is the lift force, \( \rho \) is the density, V is the flow speed of fluid and A is the relative plan area. The pressure over the airfoil varies across the surface and is dependent on the wing or spoiler geometry. The pressure on the airfoil acts normal to the surface and vary based on the surface shape, also it contributes to the lift and drag forces accordingly.

Another fundamental issue to understand is that the angle of attack. The angle of attack is a term used in aerodynamics to describe the angle at which the wing meets the air [10]. It describes the angle between the airflow direction and airfoils chord line Arising of down force generated by a wing results directly from the angle of attack, for that increasing angle of attack generating more lift and at the same time more drag. Increasing the angle of attack causes an upward or downward pressure due to air stream line defects. This phenomenon depends on the position of the longer surface and flow surface shape of an airfoil. The goal on any designer is to maximize negative lift while minimizing drag [11]. One of the critical angle specifications is the stall angle. Its used as optimal position of the angle that can produce maximum down force. The stall angle value can vary from (10 to 20 dag.) based on the airfoil type [12]. Also, the ratio of lift-to-drag is another aerodynamic design factor of the wing the lift to drag ratio (L/D) is the amount of lift force generated by airfoil compared to its drag. It is determined by dividing the lift coefficient by the drag coefficient. The drag is the price paid to obtain lift, for that drag ratio (L/D) indicates airfoil efficiency. The higher (L/D) ratios are more efficient than the lower (L/D) ratio. The higher value of this ratio is the aim factor for any wing or spoiler design [3].
3. Airfoil Characteristics

The behavior of the aerodynamic load is described as airfoil characteristics. The design target of airfoil is the application characteristics. Therefore, there are hundreds of airfoils studied by the researchers. ANCA4412 and S1223 airfoil have been studied in 2015 in a comparative study of four airfoils, S1223, S819, S8037 and S1223 RTL to investigate the effect of the angle of attack on their performance [13]. S1223 RTL airfoil founded as one of the best airfoils selected for car race. Rubel et al, in [14] studied the NACA 0015 airfoil traveling at low speed wind tunnel. They found that both drag and lift coefficient increases when the value of angle of attack is increased. Also, when the Reynolds number increases produce a limited increment in lift coefficient but it decreases drag coefficient. Biradar and Malipatil in [15] investigated the lift and drag forces of NACA 0015. The main contribution in this research is to compare the airfoil model that contains dimple on the surface with one without dimple. They found that the airfoil with dimple have less drag. Mohamed et al, in [16] investigated the GOE 387 characteristics. They found that the upper airfoil surface velocity is higher than the lower surface airfoil velocity. Also, the increases of lift force and lift coefficient observe increasing with the arising of angle of attack. Raval et al, in [17] tested NACA 0012. They presented a comparison of center of pressure variations over the airfoil surface and stall angle condition. Azmi et al, in [18] investigated three types of airfoils, NACA 2408, 2412, 2415. They founded that the increase in flap wing thickness of the airfoil will decreases the down force. Rab et al, in [2] studied the airfoil analysis code of LNV 109 A, FX 63-137, FX 64-CL 5-140 and S1223. They established the criteria for the selection of high-lift low-Reynolds-number airfoils. Harianto in [19] studied the NACA 6412 airfoil. They tested five variations of the tapper ratio. They concluded that the highest lift coefficient value was gained in the 1:1 tapper ratio variation. Aydin et al, in [20] also investigated the NACA 0015 characterization with different methods to find aerodynamic performances in terms of the lift and drag coefficients were compared to each other at different angles of attack. The results observe optimal results by applying Spalart-Almaras. Table 1 will present the finding results of the researchers.

| Table 1: C L and CD results of state-of-the-art researches |
|----------------------------------------------------------|

4
Based on the presented studies, the researcher specified the ANSYS software to test different types of airfoils to investigate the CL and CD specifications and to calculate the downforce and drag force magnitude experimentally. The main aim is to find the load behavior based on different air velocities and different angles of attack.

### 4. FLUENT Models

To specify the suitable solution method, it is necessary to prepare Computational Fluid Dynamics (CFD) model. Navier-Stokes equations considered as the best method to solve the airflow problems and investigate the changes in the flow stream. The Navier-Stokes equations observe the dynamics of a fluid flow. It is the traditional method used in all turbulence models and can be considered as a "standard" approach [7]. The Navier-Stokes equations can be derived from the three conservation equations, based on the Reynolds transport theorem [20]. The set of conservation equations below are independent of Navier-Stokes equations.
The SST model incorporates a damped cross-diffusion derivative term in the equation. These developments make the SST k-ω model more reliable and accurate for standard k-ω model [21]. The SST k-ω turbulence model has become very popular due to large production of turbulence levels in regions correlated with the effect of large normal strain with strong acceleration. For that the SST k-ω model is used as an independent method for the numerical simulation of body flow [22]. This mode have been used in this work, based on the ANSYS – CFD simulation software.

5. Methodology

5.1 Geometry Modeling and Meshing

There are a total of four models that were simulated NACA 0012, GOR493, NACA 6412 and GOE 523. The airfoils has been tested using FLUENT solver with initially compare NACA 0012 as a baseline with previous results published by Pranesh and his team in 2019 [23].
airfoil geometry were modeled in Solid Work software then imported into ANSYS software. The geometry of the models is shown in the figure 1.

The ANSYS Fluent was applied to analyze the air flow over the airfoils. Imported airfoil co-ordinates in ANSYS-Workbench to generate a two dimensional airfoil geometry and assuming that fluid flow in z-direction is negligible. Then, the fine meshing has been generated over the airfoil. The element size results observe total nodes 98961 totals elements 98386. In order to find the optimal mesh quality, the element metrics will indicate the best result. The skewness mesh metrics and orthogonal quality mesh metrics will provide the results in rage of minimum orthogonal quality of 0.15 and 0.49 for maximum skewness mesh metrics [24] [25].

5.2 Validation
In term of validation, the NACA 0012 airfoil have been chosen to test and compare with previous numerical results. The used data that had been compared obtained from Pranesh et al. presented in [23] which tested the airfoil with the condition of low Reynolds numbers to find the lift characteristics. They measured airfoil aerodynamic characteristics based on various angles of attack with free stream velocity. The comparative results present in table 2.

| angle | CL of present study | CL of (Pranesh et al, study) [23] | Error Percentage % |
|-------|---------------------|----------------------------------|--------------------|
| 4     | 0.4383              | 0.4661                           | 2.78               |
| 8     | 0.6745              | 0.6482                           | 2.63               |
| 10    | 0.652               | 0.6052                           | 4.68               |

Based on the acceptable validation results, the next step is to test the effect of airfoils based on different boundary conditions to investigate the airfoils characteristics.

### 5.3 Boundary Condition

In the present work, all the boundary conditions are applied with the 2-D dimension airfoils. The flowing fluid is considered as Air-Ideal gas and the flow velocities are applied on each of the airfoil for their comparative analysis for various flow conditions as mentioned in table 3. The inlet boundary conditions have to be specified in turbulent flows depending on the turbulence model. Typically there are three options [15]:

i. Mass flow inlet boundary condition are used in compressible flows to prescribe a mass flow rate at an inlet.
ii. Pressure inlet boundary conditions are used to define the total pressure and other scalar quantities at flow inlets
iii. Pressure outlet boundary conditions are used to define the static pressure at flow outlets
Table 3: operating boundary condition

| input                      | value          |
|----------------------------|----------------|
| 1  Velocity of flow        | 35m/s          |
| 2  Operating temperature   | 288.16 K       |
| 3  Operating pressure      | 101325 Pa      |
| 4  Model                   | SST k-ω model  |
| 5  Density of fluid        | 1.225 Kg/m³    |
| 6  Reynolds number         | $10^6$         |
| 7  Viscosity               | 1.7894e-05 kg/m-s |
| 8  Specific Heat           | 1.006.43 J/kg-k |
| 9  AOA (Angle of Attack)  | 0 degree and 20 degree |
| 10 Fluid                   | Air            |
| 11 Wall Motion             | Stationary Wall |
| 12 Initialization Method   | Hybrid Initialization |

6. Results and discussion

Four different airfoils were studied and simulated with the angle of attaching varying from 0° to 15° in increments of 5°. The GOE 652, NACA 4415, GOE 528, NACA 0012 airfoil are considered for modeling and the coordinates are developed using the standard data from the official web resource. Around the profile, the boundaries are fixed, the meshed geometry of airfoil is imported in ANSYS analyzed using the FLUENT module. Inlet velocity for the simulation is fixed as 35 m/s and turbulent flow solution called as SST k-ω model. The selected profiles are tested for finding coefficient of drag and lift and calculate the ratio of (CL/Cd) coefficient and down force and drag. The main target of these tests are to investigate the air flow behavior in different airfoils surface shapes. The analysis is performed until the values of drag and lift coefficient reaches stable in iterations at various angle of attack. The simulation results show the boundary changes in air flow and pressure of the airfoil of GOE 528 in the figure 2. The plots show that the velocity increases in the bottom of the wing, while the pressure decreases in the same surface. In the top surface, the pressure increases when the angle of attack increases.
The same behavior shown in all types of airfoils. Based on the results, the airfoil coefficients were founded and all calculations will present in next sections.

6.1 Lift and drag Coefficient

The lift coefficient involves a complex dependencies of airfoil geometry. Coefficient of lift is generated over the four airfoils and observe an increased in the lift coefficient magnitude with the increment of the angle of attack for all selected airfoils. Also, it can be observed that GOE
652 type has hair lift coefficient as shown in figure 3. The other values were NACA 4415, GOE 528 and NACA 0012 respectively.

![Figure 3: the coefficients result](image)

Similar behavior can be seen in drag coefficient. It is clear that, GOE 652 produces the highest amount of drag coefficient whereas NACA 4415, GOE 528 and NACA 0012 produces lowest amount of drag coefficient table 4 presents the variations of lift and drag coefficients at different angles of attack for these four airfoils.

| Alpha | GOE 652 CD | CL | NACA 4415 CD | CL | GOE 528 CD | CL | naca 0012 CD | CL |
|-------|------------|----|--------------|----|------------|----|--------------|----|
| 0     | 0.0132     | 1.3170 | 0.0085       | 0.434 | 0.0104     | 0.5993 | 0.0000       | 0.0000 |
| 5     | 0.0212     | 1.8230 | 0.0096       | 0.965 | 0.0115     | 1.0979 | 0.0082       | 0.4504 |
| 7     | 0.0249     | 1.9310 | 0.0137       | 1.161 | 0.0131     | 1.2917 | 0.0107       | 0.8657 |
| 9     | 0.0332     | 2.1010 | 0.0241       | 1.219 | 0.0195     | 1.4948 | 0.0150       | 0.9557 |
| 11    | 0.0469     | 2.2530 | 0.0332       | 1.364 | 0.0303     | 1.6934 | 0.0324       | 1.2095 |
| 15    | 0.0571     | 2.3320 | 0.0650       | 1.491 | 0.0735     | 1.8921 | 0.0396       | 1.3470 |
As shown in the table 4, the values of coefficients in all cases are increased from angle of attack 0 to 15, but there is a slight increment in NACA 4415 and GOE 528 drag coefficients after angle 13°. The ratio of lift to drag is another important airfoil characteristic that obtained from the simulation results. Based on the numerical results founded from ANSYS, the lift to drag ratio has been calculated as shown in table 5.

Table 5: numerical results of CL/CD ratio

| angle of attack | naca 0012 | GOE 528 | NACA 4415 | GOE 652 |
|----------------|-----------|---------|-----------|---------|
| 0              | 0         | 57.79171| 51.05882  | 99.77273|
| 5              | 54.86472  | 95.63589| 100.8359  | 85.99057|
| 7              | 81.05805  | 98.45274| 84.74453  | 77.5502 |
| 9              | 63.6285   | 76.61712| 50.58091  | 63.28313|
| 11             | 37.33025  | 55.90624| 41.08434  | 48.03838|
| 15             | 34.01515  | 25.74636| 22.93846  | 40.84063|

The results observe an increment in NACA 4415, GOE 528 and NACA 0012 lift to drag ratio from zero degree to angles (5 to 7) degrees. Then the ratio values decrease. In case of GOE 652 airfoil, the lift to drag ratio decreases in all cases. The airfoils down forces are another task in this work. The numerical simulation presents the down force values as shown in table 6.

Table 6: numerical results of down forces

| angle of attack | NACA 0012 (N) | GOE 528 (N) | NACA 4415 (N) | GOE 652 (N) |
|----------------|---------------|-------------|---------------|-------------|
| 0              | 513.2138      | 373.7843    | 1076.894      |             |
| 5              | 368.2783      | 941.7065    | 1490.644      |             |
| 7              | 707.8721      | 1108.662    | 984.41        | 1578.955    |
| 9              | 781.4639      | 1281.105    | 1047.008      | 1717.961    |
| 11             | 988.993       | 1451.062    | 1167.486      | 1842.25     |
| 15             | 1101.425      | 1622.133    | 1283.678      | 1906.847    |

The results shown the same response of force growth. The down force increases with all cases when the angle of attack increases for all airfoil models. The results observe a clear relationship between the airfoil surface curve shapes with lift and drag coefficients. Based on the results, the airfoil surface geometry investigated. The researcher measure the airfoil internal curve by
trapezoidal rule which represent a method to evaluate the area of the airfoil surface curve by dividing the total area into smaller trapezoids. The results shown in table 7.

Table 7: numerical results of airfoil surface area

| airfoil   | Area (m) |
|-----------|----------|
| NACA 0012 | 0.0006   |
| GOE 528   | 0.2957   |
| NACA4415  | 0.6546   |
| GOE 652   | 1.0769   |

The relationship between the curve area of airfoil and the down force magnitude is shown in the figure 4:

![Figure 4: the relationship between the airfoil down force and the geometrical surface area](image)

The present plot can facilitate the design processes for choosing the suitable spoiler or wing to add it to the car. It observes the big increase of the down force after the area value of (0.83) of the airfoil surface curve. Also, it gives a significant view to the designer to estimate the down force effect when add a wing or spoiler.
7. Conclusion

In this paper, analysis of four airfoils has been carried out with different angles of attack by using ANSYS fluent, the used model was Model was SST k-ω and the velocity was 35 m/s. the spoiler or wing selection process, it is most desirable to choose a suitable airfoil to achieve an optimal lift to drag ration. The amount of down force can be generated based on specified boundary conditions. For this purpose, Lift-to-Drag ratio and coefficient of lift and drag were the most important taken values. They present the aerodynamic design specification of the airfoil and guide to measure the differences in aerodynamic effects. The trapezoidal of surface area has been calculated to find out the relationship between the aerodynamic design specification and the wing surface shape and area. From the analysis, it can be concluded that airfoil GOE 652 has hair lift coefficient specifications. The other values of NACA 4415, GOE 528 and NACA 0012 present the maximum drag-to-lift ratio, maximum value of coefficient of lift and the second lowest value of drag coefficient at an angle with range (from 5 to 7) degree. Based on these results, the down force specified and found of GOE 652 which has the most effective geometry among the four airfoils.

Acknowledgments
The author would like to thank Mustansiriyah University (www.uomustansiriyah.edu.iq) Baghdad-Iraq for its support in the present work.

References

[1] N. Rastogi, S. Shetty, and B. Ashok, “Analysing the effects of air flow on a formula prototype vehicle to optimize its performance,” IOP Conf. Ser. Mater. Sci. Eng., vol. 263, no. 6, 2017, doi: 10.1088/1757-899X/263/6/062033.

[2] M. F. Ur Rab and M. J. Alam, “Comparative analysis of high-lift airfoils for motorsports applications,” WIT Trans. Eng. Sci., vol. 120, no. July, pp. 17–183, 2018, doi: 10.2495/AFM180171.

[3] Sumit Sharma, “An Aerodynamic Comparative Analysis of Airfoils for Low-Speed Aircrafts,” Int. J. Eng. Res., vol. V5, no. 11, pp. 525–529, 2016, doi: 10.17577/ijertv5is110361.

[4] B. N. Devaiah and S. Umesh, “Enhancement of Aerodynamic Performance of a Formula-1 Race Car Using Add-on Devices,” sasTech J., vol. 12, no. 1, pp. 72–79, 2013.
[5] P. Gowdhaman, K. Antonyraj, and V. Annamalai, “An effective approach on physical and dielectric properties of PZT-PVDF composites,” Int. J. Adv. Sci. Res., vol. 1, no. 08, pp. 322–328, 2015, doi: 10.7439/ijasr.

[6] S. Kandwal and S. Singh, “Computational Fluid Dynamics Study Of Fluid Flow And Aerodynamic Forces On An Airfoil,” Int. J. Eng. Res. Technol., vol. 1, no. 7, pp. 1–8, 2012.

[7] A. A. Matyushenko, E. V. Kotov, and A. V. Garbaruk, “Calculations of flow around airfoils using two-dimensional RANS: an analysis of the reduction in accuracy,” St. Petersbg. Polytech. Univ. J. Phys. Math., vol. 3, no. 1, pp. 15–21, 2017, doi: 10.1016/j.spjjpm.2017.03.004.

[8] N. M. Triet, N. N. Viet, and P. M. Thang, “Aerodynamic Analysis of Aircraft Wing,” VNU J. Sci. Math. – Phys., vol. 31, no. 2, pp. 68–75, 2015.

[9] S. R. Nair, “Study and Comparison of Aerodynamic Forces on an Aircraft and Suspension Bridge,” IOSR J. Eng., vol. 4, no. 4, pp. 43–46, 2014, doi: 10.9790/3021-04454346.

[10] S. Hiremath and A. S. Malipatil, “CFD Simulations of Aircraft Body with Different Angle of Attack and Velocity,” Int. J. Innov. Res. Sci. Eng. Technol., vol. 03, no. 10, pp. 16965–16972, 2014, doi: 10.15680/ijirset.2014.0310077.

[11] V. Kshirsagar and J. V. Chopade, “Aerodynamics of High Performance Vehicles,” Int. Res. J. Eng. Technol., vol. 5, no. 3, pp. 2182–2186, 2018.

[12] M. M. Rubiat Mustak, Md. Harun-Or-Rashid Molla, “Improvement of Aerodynamic Characteristics of an Airfoil by Surface Modification,” Am. J. Eng. Res., vol. 6, no. 3, pp. 07–14, 2017, doi: 10.6084/m9.figshare.4868774.

[13] J. Singh, J. Singh, A. Singh, A. Rana, and A. Dahiya, “Study of NACA 4412 and Selig 1223 airfoils through computational fluid dynamics,” vol. 2, no. 6, 2015.

[14] R. Islam Rubel, M. Kamal Uddin, M. Zahidul Islam, and M. Rokunuzzaman, “Numerical and Experimental Investigation of Aerodynamics Characteristics of NACA 0015 Aerofoil,” Int. J. Eng. Technol. IJET, vol. 2, no. 4, pp. 132–141, 2017, doi: 10.19072/ijet.280499.
[15] S. Biradar and A. S. Malipatil, “CFD Analysis of Dimple Effect on Airfoil Naca,” vol. 5, no. IX, pp. 369–374, 2017.

[16] M. A. F. Kandil and A. O. Elnady, “CFD Analysis of GOE 387 Airfoil,” vol. 14, no. 5, pp. 80–85, 2017, doi: 10.9790/1684-1405018085.

[17] N. P. Raval, M. Malay, and L. Jitesh, “CFD Analysis of NACA0012 Aerofoil and Evaluation of Stall Condition,” vol. 4, no. 4, pp. 57–60, 2017.

[18] A. R. S. Azmi, A. Sapit, A. N. Mohammed, M. A. Razali, A. Sadikin, and N. Nordin, “Study on airflow characteristics of rear wing of F1 car,” IOP Conf. Ser. Mater. Sci. Eng., vol. 243, no. 1, 2017, doi: 10.1088/1757-899X/243/1/012030.

[19] Y. H. Irawan, E. Yawara, and H. Bakhtiar, “Effect of Tapper Ratio on a Car Rear Spoiler Performance,” vol. 4, no. 1, pp. 1–14, 2019.

[20] N. Aydın, M. E. Çalışkan, and İ. Karagöz, “Numerical Simulation of Flow Over Different Types of Airfoils,” no. January, 2020.

[21] K. Kurec, M. Remer, J. Broniszewski, P. Bibik, S. Tudruj, and J. Piechna, “Advanced Modeling and Simulation of Vehicle Active Aerodynamic Safety,” J. Adv. Transp., vol. 2019, 2019, doi: 10.1155/2019/7308590.

[22] A. K. Saraf, M. P. Singh, and T. Chouhan, “A Review on Aerodynamic Behavior of Airfoil when Surface Modified,” vol. 7, no. 3, pp. 516–519, 2016.

[23] C. Pranesh, M. Sivapragasam, M. D. Deshpande, and H. K. Narahari, “Negative lift characteristics of NACA 0012 aerofoil at low Reynolds numbers,” Sadhana - Acad. Proc. Eng. Sci., vol. 44, no. 1, pp. 1–6, 2019, doi: 10.1007/s12046-018-1008-6.

[24] B. Fabritius and G. Tabor, “Improving the quality of finite volume meshes through genetic optimization,” Eng. Compute., vol. 32, no. 3, pp. 425–440, 2016, doi: 10.1007/s00366-015-0423-0.

[25] E. S. Mohammed, “Study of the Possibility of Achieving the Same Per-Port Outflow in a Dividing Manifold,” vol. 12, no. 2, pp. 329–347, 2017.