Published Online October 2022

Dept. of Mech. Engg.
CUET

Mechanical Engineering Research Journal

Vol. 12, pp. 54-62, 2020

ISSN: 1990-5491

NUMERICAL AND EXPERIMENTAL INVESTIGATIONS ON THE INFLUENCE OF SURFACE MODIFICATIONS INTEGRATED WITH VORTEX GENERATORS ON NACA0010 AIRFOIL

Srijan Dasgupta[1], Kazi Afzalur Rahman[2*], Md. Kawsar Sajib[3], Md. Ibrahim Khalil Tanim[4]

[1]Universitй de Lorraine, Nancy, FRANCE
[2,4]Department of Mechanical Engineering, Chittagong University of Engineering & Technology, BANGLADESH
[3]University of Liиge, Liиge, BELGIUM

ABSTRACT

Vortex generators and surface modifications are done to reduce the wake region and increase the aerodynamics efficiencies of an airfoil. The performance test of NACA0010 airfoil with various types of surface modifications integrated with vortex generators and comparison to a conventional rigid body airfoil is presented in this paper. The characteristics of airfoils were tested using numerical analysis and was validated using AF100 subsonic wind tunnel testing. Both the computations and experimentations were conducted at constant chord Reynolds number of 143,500. The test data shows the comparison between conventional NACA0010 airfoil with three types of surface modifications integrated with vortex generators at an angle of attack of zero degree. This study suggests that the airfoil with strips and vortex generators tend to increase lift to drag ratio most effectively. However, addition of vortex generators and surface modification was less effective in low angle of attack.

Keywords: *Vortex generators, Airfoil modifications, Serration, Computational fluid dynamics, lift, drag.*

NOMENCLATURE

C_L	= Coefficient of lift
C_D	= Coefficient of drag
C	= Cord Length
V	= Velocity of air
L	= Lift force
D	= Drag Force
K	= Turbulent kinetic energy
ε	= Turbulent dissipation

1. INTRODUCTION

Airfoils are made by creating effective curvatures, which gives the wing like structure on various aircrafts and other technologies. The Wright brothers were the first persons to do research on structures that we call airfoils [1]. The discovery of airfoil was made possible by doing the cross-sectional study of birds and the aerodynamic behaviors like lift, drag, boundary layer formation etc. around bird wings. Since then, the aviation industry is just blooming, yet the airfoils are yet to show better aerodynamic behaviors like the natural flyers in case of agility, speed, agility and so on [2]. Various surface modifications and extensions on airfoil models including various cuts, dimples and vortex generators are used to investigate various aerodynamic properties and discover improvements. The vortex generators have a characteristic of delaying flow separation, which decreases wake and pressure drag. Decreasing drag is one of the main concerns in most aerodynamic cases. The dimples and surface modifications are some sorts of vortex generators themselves in case of functions. Better lift and lesser drag investigations are necessary to decrease fuel consumption and maximize technological advantages. Usually, the more streamlined a body is, the less wake and pressure drag is generated. The flow tends to be a streamlined flow if the flow separations is less or delayed. The function of vortex generators is creating vortices, that tends to create turbulence and delay flow separations [3]. Also, various surface modifications such as dimples creates perturbation of flow around the dimples which creates separation bubbles. Usually, flow is accelerated around the dimpled surfaces to cause a transition from laminar to turbulent flow. This transition to turbulent flow

Received : *28 March 2021*
Revised : *12 January 2022*
Accepted : *20 April 2022*

**Corresponding Author:*
E-mail: afzal@cuet.ac.bd

causes the flow separation at the trailing edge to delay, delaying the pressure drag significantly [4]. Similar observations can also be noticed in airfoils. For example, using protuberances along the leading edge of NACA 2412, it is possible to attain better performance from the baseline [5].

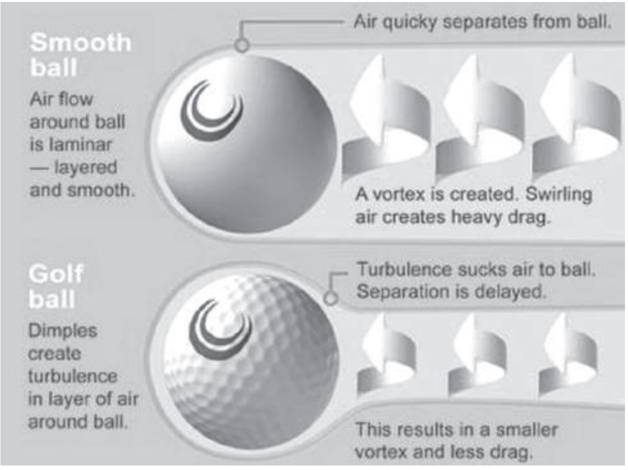


Fig. 1: Delay of flow separation due to dimples on a golf ball [6].

However, the introduction of surface modifications may increase skin friction drag. Along with dimples and strips, trailing edge serration was also used for the purpose of investigation. Serrations are usually provided to decrease the level of noise by decreasing the trailing edge wake turbulence formation [7]. However, different shapes of serration can affect lift and drag ratio as well, as serrations tend to increase lift to drag ratio significantly and square serration provides the best performance rather than triangular serrated flap [8]. On a similar note, The quantitative and qualitative results on the efficacy of the proposed triangular counter rotating micro vortex generators show that the deployment of micro-vortex generator reduces the boundary layer thickness in the flow past them and delays separation [9].

The aerodynamic performance of new designed airfoils can be improved with appropriate installation and optimization of VGs. It is usually suggested that VGs be placed at x > 20%c to avoid decrease of lift coefficient at small angles of attack for the tested airfoils [10]. The aerodynamic investigations like these can be done on numerous possible systems which includes numerical analysis like computational fluid dynamics (CFD) and experimental strategies like the wind tunnel testing. Computational fluid dynamics (CFD) is a way to achieving computational physics, opening a new branch of fluid dynamics. Numerical calculations and algorithms are used to get solutions and to anatomize problems involving fluid flows of specific conditions. With the introduction of high-speed supercomputers, achievement of better solutions was made a lot easier than before [11]. This paper shows a comparison between the data achieved from numerical analysis and experimental setup to better understand the aerodynamics of the integration of vortex generators with surface modifications.

2. MATERIALS AND METHODS:

All three designs were integrated with trailing edge vortex generators.

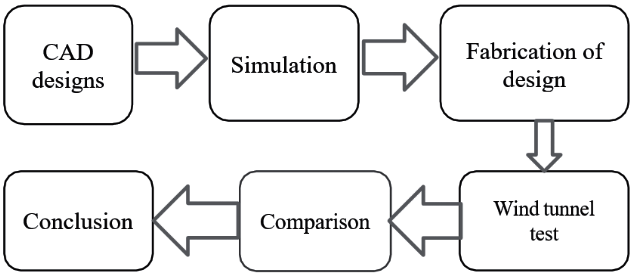


Fig. 2: Flow chart of analysis procedure.

The numerical analysis was done using Ansys student version and later fabricated specimens were tested in the subsonic wind tunnel.

2.1. Airfoil Selection

For this paper, NACA 0010 airfoil was selected for investigation. The National Advisory

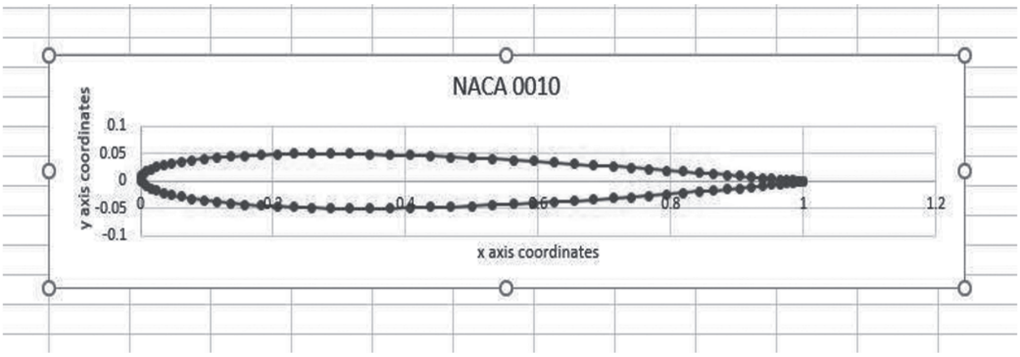


Fig. 3: NACA 0010 coordinate plotting.

Committee for Aeronautics (NACA) develops various shapes and curvatures for various applications. The four-digit value actually indicates the airfoil specifications. NACA 0010 is a symmetric airfoil, thus it has to camber. The maximum thickness of the model is at 0.10C.

2.2 Numerical Solution

The models were designed in the 3D modelling software named SOLIDWORKS. The necessary modifications and vortex generators were then added. The airfoil that was selected for the purpose of analysis was NACA0010 airfoil. The material that was chosen for the analysis was Gamari wood. The chord length was selected to be 150mm and the span was 300mm. Modifications on the airfoil was done using slots, strips and vortex generators as shown in the figure 04.

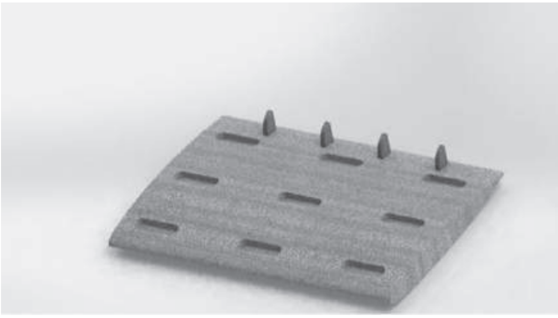


Fig. 4: Airfoil with slots and vortex generators

Fig. 5: NACA 0010 airfoil with strips and vortex generators

Fig. 5: NACA 0010 Airfoil with serrations and vortex generators

The vortex generators were made out of PVC (Poly-Vinyl Chloride) sheets and the type of the vortex generators was Gothic VGs. The height of the vortex generators was 12 mm and the base was of 10mm.

2.2.2 Mesh Generation

The geometries were imported to the Ansys design modeler, where boolean was created to make a rectangular enclosure of fluid region around the airfoil. To complete this simulation, the adaptive mesh refinement technique was used. refinement or AMR, is in fact a technique of adjusting the precision of a solution within certain delicate or turbulent simulation areas, dynamically or during the calculation of the solution. In the sizing option of the mesh, the element sizing was set to fine and the smoothing option was set to high. Other default options were unaltered and mesh were generated on each of the designs.

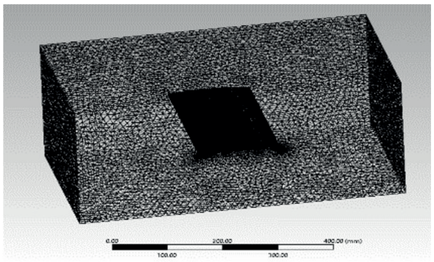


Fig. 6: Meshing done on ANSYS Fluent

2.2.3 Governing equations

The governing equations that are embedded into the Ansys fluent are Continuity equation and three dimensional Navier-Stokes equations. As the flow was assumed as an incompressible flow, the energy equations were not necessary for this analysis.

Due to incompressible flow assumption, the continuity equation can be written as,

$$\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)=0$$

The Navier-Stokes equation is the partial differential form of the conservation of momentum, which is the most popular set of equations in fluid mechanics [12]. These equations can be written as,

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

However, during numerical analysis, Reynolds averaged Navier-Stokes (RANS) equations are used using finite volume method. For solving RANS equations, k-epsilon turbulent model consisting of two equations are used. It is widely used for confined experiments, where the pressure gradient is relatively small, which was the case for our experimental setup as well.

The first equation is for the turbulent kinetic energy k, which is,

$$\frac{\partial(\rho K)}{} + \frac{\partial(\rho k u_i)}{} = \frac{\partial}{\partial x_i}\left|\frac{\mu}{\sigma_k}\frac{\partial k}{\partial x_j}\right| + 2\mu_k E_{i,j}.\,E_{i,j} - \rho\varepsilon$$

The second equation is for the dissipation,

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i}\left|\frac{\mu}{\sigma_s}\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1s}\frac{\varepsilon}{K}2\mu_t E_{i,j}.\,E_{i,j} - \rho\frac{\varepsilon^2}{K}C_{2s}$$

There are a number of constants that are used in this turbulent model, which has some value that has been found by numerous number of iterations of data fittings [13].

2.2.3 Boundary Conditions

The solver type in the Ansys fluent was set as pressure-based solver due to incompressible flow assumption, with steady time and absolute velocity formation being selected. The gravity option was not turned on in this analysis. For the inlet boundary condition, inlet velocity was set to 15 m/s and the outlet gage pressure was set to zero. No slip conditions on the wall boundary conditions were set. One wall was set as "symmetry" boundary condition. In the solution methods, "coupled" scheme was selected along with second order calculations in pressure, momentum, turbulent kinetic energy and turbulent dissipation rate. In summary, the simulation was done by using the following parameters:

- The fluid will be assumed as incompressible.
- Fluid type: Air.
- Temperature: 301K.
- Pressure: 101325 Pa.
- Air density: 1.17 kg/m^3.
- Velocity for air: 15 m/s.
- Test object material: Wood (Gamari).
- Material density: 510 kg/m^3.

2.3. Airfoil prototypes

The designs that were generated using the software named SOLIDWORKS were taken to the wood shop with necessary dimensions for fabricating the airfoils. The vortex generators were made by cutting PVC sheets out to the desired dimensions and were joined together with the airfoils with adhesives.

2.4 Wind Tunnel Testing

The simulation results were based on various ideal assumptions. But in real life problems, fluid particles do not follow the idea conditions of the theoretical fluid mechanics. The AF100 subsonic wind tunnel provides a compact and practical open circuit wind tunnel in which low speed aerodynamic effects are studied.

The experimental investigation was done in the fluid lab of Chittagong University of Engineering and Technology. The wind tunnel used here belongs to a closed working segment, open type return suction. It is backed with a lockable castor on a tubular steel frame, which makes the device very mobile. Air enters the tunnel through a cone intended aerodynamically to accelerate the air linearly. It then enters the working section and goes through a grill before entering through a diffuser and through the axial fan with variable velocity. The air leaves the fan, passes through the unit of silencer and then returns to the atmosphere.

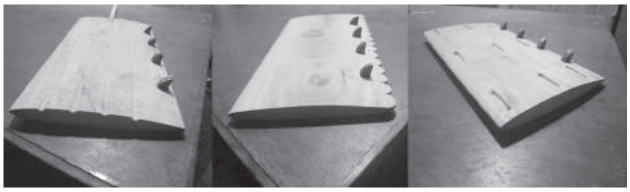


Fig. 7: Fabricated NACA0010 airfoil wings with surface modifications and vortex generators

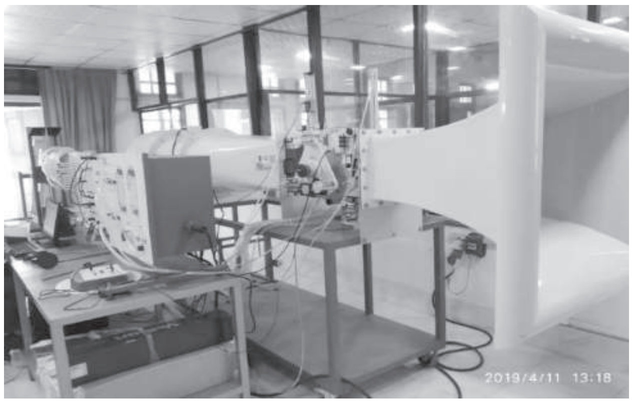


Fig. 8: AF100 subsonic wind tunnel in CUET lab

Using the VDAS software (Versatile Data Acquisition System) and simple USB links, the digital system was set up for the data collection center to collect, calculate and export the lift

Fig. 9: Working section of the wind tunnel

and drag coefficients directly by using the manometers and the AFA3 three component balance of the AF100 subsonic wind tunnel [14].

3. DATA TABLES AND FIGURES

The simulation data were compared to the experimental data to validate the results of the simulation. The simulation and the wind tunnel testing were done using similar environmental conditions. The calculated data of the simulation and the wind tunnel test showed variation in the values of lift coefficient, drag coefficient and lift to drag ratio. The conventional NACA0010 without any modifications and VGs was also investigated for the comparing this airfoil with the airfoils with modifications and VGs. The data tables with error calculations with respect to the wind tunnel data are given.

Table 1: Experimental and Simulation data of conventional airfoil

Experimental			Simulation			Error	
C_L	C_D	C_L/C_D	C_L	C_D	C_L/C_D	% of Error in C_L	% of Error in C_D
0.0165	0.076	0.22	0.0031	0.019	0.16	81.2	75

conventional NACA0010 without any modifications and VGs was also investigated for the comparing this airfoil with the airfoils with modifications and VGs. The data tables with error calculations with respect to the wind tunnel data are given.

Table 2: Experimental and simulation data of airfoil with strips vortex generators

Experimental			Simulation			Error	
C_L	C_D	C_L/C_D	C_L	C_D	C_L/C_D	% of Error in C_L	% of Error in C_D
0.198	0.0976	2.03	0.0364	0.0264	1.38	81.6	73

There was a limitation in mesh refinement in the simulation study procedure. Also, due to slight inaccuracies in dimensions of fabrication properties, there was noticeable differences

Table 3: Experimental and simulation data of airfoil with serrations vortex generators

Experimental			Simulation			Error	
C_L	C_D	C_L/C_D	C_L	C_D	C_L/C_D	% of Error in C_L	% of Error in C_D
0.0187	0.088	0.21	0.00387	0.0258	0.15	79	70.1

between the simulation results and wind tunnel test results. However, the results followed near similar patterns of changes in lift and drag coefficients in both operations. The side-by- side comparison of the values of the lift coefficient, drag coefficient and lift to drag ratio with surface modification integrated with vortex generators along with the values of the conventional NACA0010 are shown in the bar charts. patterns of changes in lift and drag coefficients in both operations. The side-by-side comparison of the values of the lift coefficient, drag coefficient and lift to drag ratio with surface modification integrated with vortex generators along with the values of the conventional NACA0010 are shown in the bar charts.

Experimental			Simulation			Error	
C_L	C_D	C_L/C_D	C_L	C_D	C_L/C_D	% of Error in C_L	% of Error in C_D
0.15	0.081	1.85	0.028	0.026	1.08	81	68

Table 4: Experimental and simulation data of airfoil with slots vortex generators

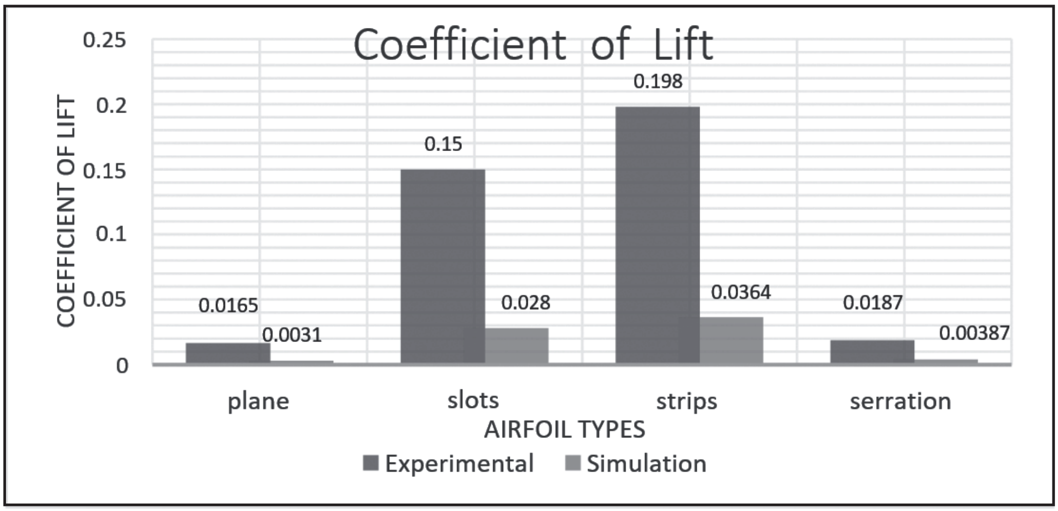


Fig.10: Comparison of the coefficient of lift

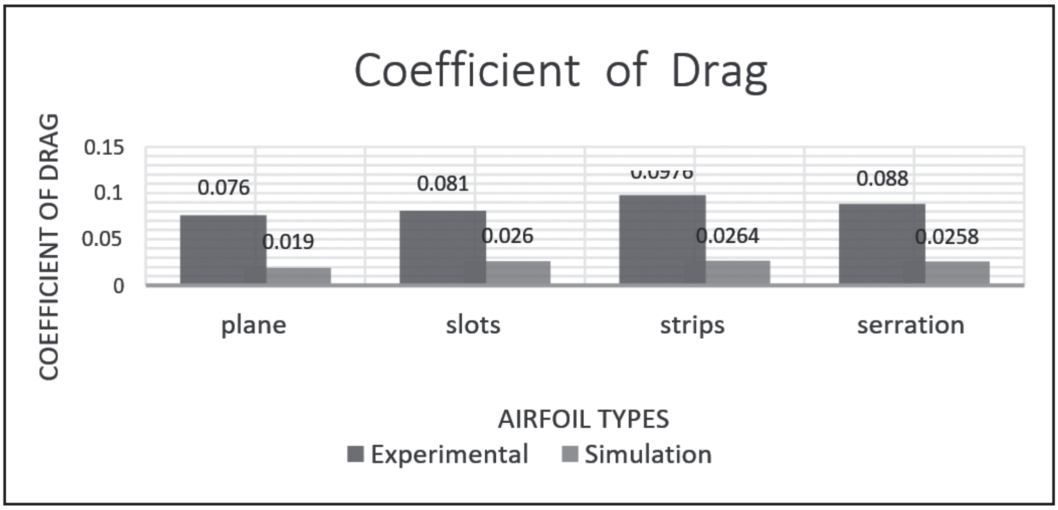


Fig. 11: Comparison of the coefficient of drag.

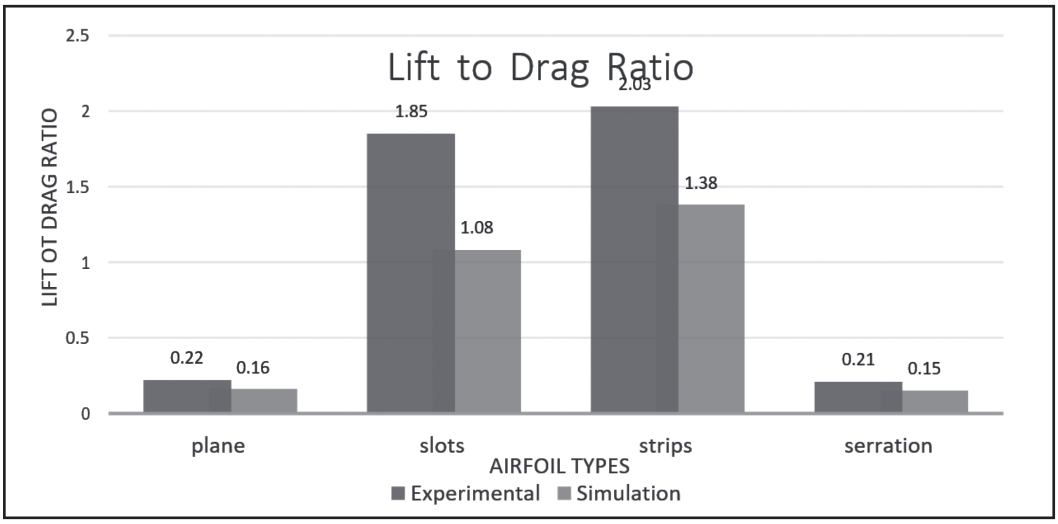
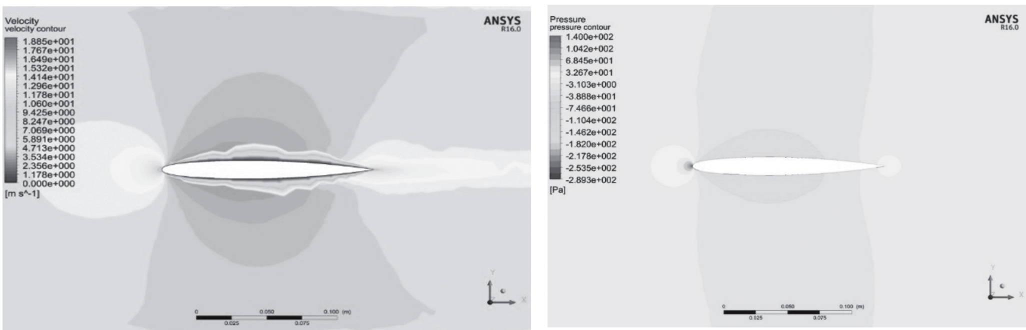


Fig.12: Comparison of the lift to drag ratio

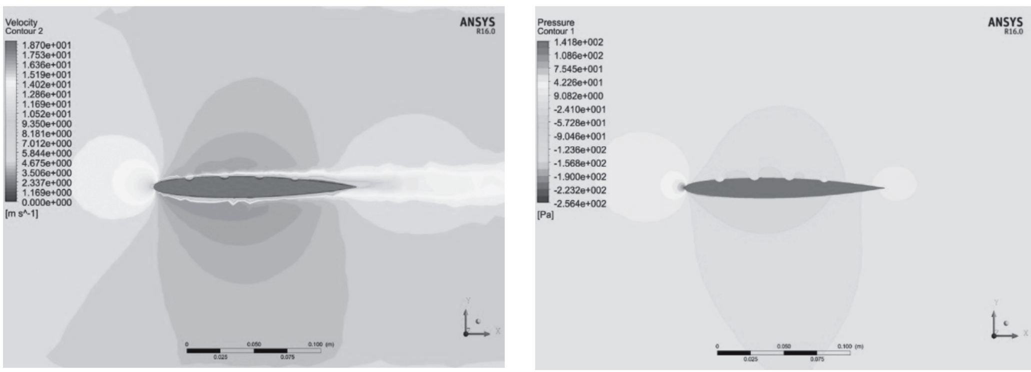
4. INVESTIGATIONS AND DISCUSSION

The comparison shown in the data tables were done to validate the CFD results and to understand the extent to which the simulation data were accurately analyzed with respect to the real-world fluid behaviors. However, the trend of error between the simulation and experimental data was quite similar.

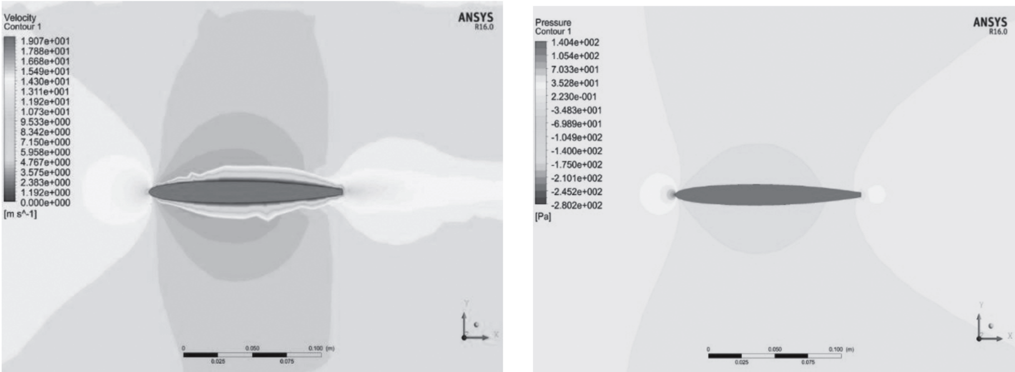
In the case of the coefficient of lift, it showed that the airfoil with strips and vortex generators showed the highest value. The conventional airfoil along with the airfoil with serrations and vortex generators showed the least value of coefficient of lift. This can be explained the velocity contours shows a symmetric pattern in both conventional airfoil and airfoil with



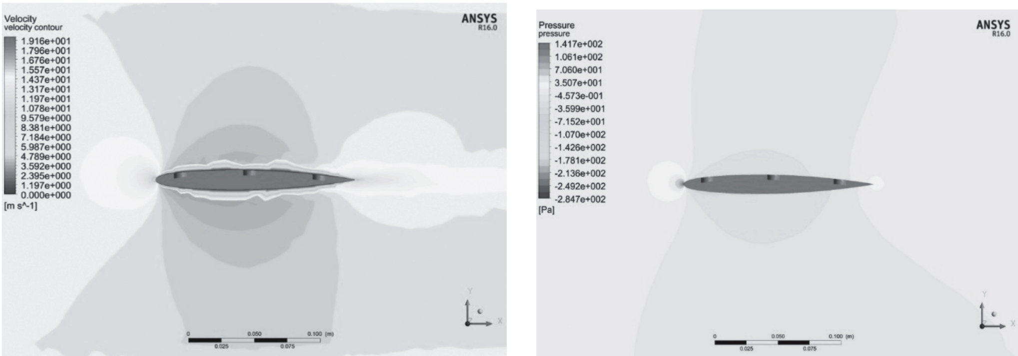
Figs. 13 (a) Conventional NACA0010 airfoil,



Figs. (b) Airfoil with strips and vortex generators



Figs. 13(c) Airfoil with serration and vortex generators



Figs. 13(d) Airfoil with slots and vortex generators

serrations and vortex generators. In case coefficient of drag, the conventional airfoil and the airfoil with slots showed the lowest values. The reason for this kind of result can also be observed in the velocity contours of the simulation data. The trailing edge wake formation is observed to be less on velocity contours in case of conventional airfoils and airfoil with slots and VGs. However, the wake formation is largest in case of the airfoil with strips and VGs, thus it showed the highest value of coefficient of drag.

5. CONCLUSION

The results obtained from both the experimentation and the numerical analysis differed to a certain extent. The experimental data showed much higher value of lift and drag coefficients compared to the simulated converged data. This difference in the experimental and simulated data was caused due to the difference of the actual and computational environmental variables that were set before proceeding with the analysis.

Results from CFD and wind tunnel experimentation of subsonic flows can be categorized into the following major research outputs:

· The coefficient of lift (C_L) increases with each surface modifications integrated with vortex generators on the airfoil. The airfoil with strips and vortex generators showed the highest value of coefficient of lift in both experiment and simulation. The increase in the lift coefficient on the airfoil with serration and vortex generators was negligibly small.

· The coefficient of drag (C_D) increases with each surface modifications integrated with vortex generators on the airfoil. The analysis was conducted at an angle of attack of zero degree. Usually, vortex generators become effective at high angle of attacks by reducing the pressure drag. Thus, it can be said that these modifications rather increase pressure drag at an angle of attack of zero degree.

· The airfoils with strips and slots with gothic vortex generators gave the highest values lift to drag ratios. The modification with serration was not that effective in increasing this value.

• Even though in all surface modifications, both the lift and drag values were increased, but considering the increase in lift with respect to the increase in drag, the surface modifications with strips integrated with vortex generator showed better advantage over other airfoil modifications at an angle of attack of zero degrees.

As a concluding remark, it can be said that better results could have been generated if the analysis was conducted on high Mach numbers. Also, previous investigations suggest that vortex generators are more effective at high angle of attacks [15]. Also, because of the limitations of the computational power available, the mesh constructions were simplified, thus limiting the accuracy of the analysis. Also, the source of error in the experimental data was due to the slightly rough surfaces of wooden airfoils, vortex generators made out of PVC sheets and unnoticeable inaccuracies of fabrication.

REFERENCES

[1] "Development of Aviation Technology," 2017. [Online]. Available: http://www.century-of-flight.net / Aviation history/evolution of technology/ Airfoils.htm (March 28, 2019)

[2] A. Hadhazy, "In the Future, Airplane Wings Will Flap," Discover Magazine, 2013.

[3] S. S. Mahamuni, "International Journal of Aerospace and Mechanical Engineering A Review on Study of Aerodynamic Characteristics of Dimple Effect on Wing International Journal of Aerospace and Mechanical Engineering," vol. 2, no. 4, pp. 18–21, 2015.

[4] E. Livya, G. Anitha, and P. Valli, "Aerodynamic Analysis of Dimple Effect on Aircraft Wing," vol. 9, no. 2, pp. 350–353, 2015.

[5] C. Paper, M. Masud, N. U. Hasan, and A. M. E. Arefin, "Design Modification of Airfoil by Integrating Sinusoidal Leading Edge and Dimpled Surface Design modification of airfoil by integrating sinusoidal leading edge and dimpledsurface View online : http://dx.doi.org/10.1063/1.4984677 View Table of Contents : http://aip.scitation.org/toc/apc/1851/1 Published by the American Institute of Physics," no. March 2018, 2016.

[6] "Dimples." [Online]. Available: https://polymerracingproducts.com/documentation/dimples. (July 24, 2019)

[7] X. Liu, H. K. Jawahar, and M. Azarpeyvand, "Development of Airfoils with Serrated Trailing Edges. In 22nd AIAA CEAS Aeroacoustics Conference [AIAA 2016-2817] American Institute of Aeronautics and Astronautics Inc. (AIAA). https://doi.org/10.2514/6.2016-Wake Development of Airfoils with Serrated Trailing Edges," 2016.

[8] S. K. Saha and M. Alam, "ICMERE2017-PI-129 NUMERICAL AND EXPERIMENTAL INVESTIGATION OF THE INFLUENCE OF," vol. 2017, pp. 18–20, 2017.

[9] X. Dong and Y. Chen, "CFD Studies on Triangular Micro-Vortex Generators in Flow Control."

[10] L. Zhang, X. Li, K. Yang, and D. Xue, "Journal of Wind Engineering Effects of vortex generators on aerodynamic performance of thick wind turbine airfoils," Jnl. Wind Eng. Ind. Aerodyn., vol. 156, pp. 84–92, 2016.

[11] "Computational Fluid Dynamics," 2016. [Online]. Available: https://en.wikipedia.org/wiki/Computational_fluid_dynamics. (March 30, 2019)

[12] P. j. Pritchard and J. c. Leylegian, Introduction to fluid mechanics, 8th ed. .

[13] "k-epsion turbulent model." [Online]. Available: https://en.wikipedia.org/wiki/K-epsilon_turbulence_model. (July 01, 2019).

[14] T. Academia, "Versatile Data Acquisition System."

[15] J. S. Thurairaj, "Numerical Analysis of Drag Reduction Method Using Vortex Generator on Symmetric Aerofoil," no. May, 2016.