CFD Letters 16, Issue 11 (2024) 1-16

SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

CFD Analysis of Counter-Rotating Vane-Type Wing Vortex Generator for Regional Aircraft

Gunawan Wijiatmoko[1,2,*], Eflita Yohana[1], Putro Adi Nugroho[1], Mohammad Tauviqirrahman[1], Ivransa Zuhdi Pane[2]

[1] Department of Mechanical Engineering, Faculty of Engineering, Diponegoro University, Semarang, Indonesia
[2] Laboratorium Aerodinamika, Aeroelastika, dan Aeroakustika, LA3-BRIN, Tangerang Selatan, Indonesia

ARTICLE INFO

Article history:
Received 1 December 2023
Received in revised form 5 January 2024
Accepted 8 February 2024
Available online 30 June 2024

Keywords:
Wing Vortex Generator; Numerical Simulation; Stall; CFD

ABSTRACT

Vortex generator is a component that has a significant impact on aircraft performance. The function of the vortex generator is to create vortices that can optimize the aerodynamic performance of aircraft wings by avoiding air flow separation and increasing lift at high angle of attack. Vortex generator can provide increased lift during take-off and landing due to the increased wing angle of attack. Although the use of vortex generator can be carried out using an experimental approach, a computational fluid dynamic approach to determine the influence of geometric parameters and placement of the vortex generator needs to be carried out. The aim of this research is to determine the effect of parameters like placement on the wing chord, height of the boundary layer, length, shape, angle of incidence and distance between pairs on the lift and drag. The model used as a computational fluid dynamic calculation model is the Spalart Allmaras transient model. As a result, vortex generator does not always have a good effect on aerodynamics. All configurations have a negative influence on the lift and drag values, but the flow separation phenomenon can be reduced significantly. Of all the configurations, the best configuration is obtained by exhibiting an ogive shape, positioned at 13.8% of the chord length, set at a 13^{o} angle of incidence. The vortex generator should have a height closely matching the boundary layer, with a length 6.5 times the height and a pair spacing of 6.7 times the height.

1. Introduction

Regional aircraft are a type of aircraft that serves short to medium flight routes. Usually, connecting small or regional cities with a small passenger capacity for regional airlines or short flights [1]. Installation of vortex generator (VG) on regional aircraft is necessary to improve aerodynamic performance. In stall conditions, the lift drop and drag heaps due to flow separation [2]. VG is used to control air flow around aircraft wings, reduce the risk of air flow separation (stall), and increase lift [3]. This helps aircraft to be able to take off and land on the shorter runways that regional aircraft

* *Corresponding author.*
E-mail address: gunawan.wijiatmoko@brin.go.id (Gunawan Wijiatmoko)

https://doi.org/10.37934/cfdl.16.11.116

encounter at small airports, as well as reducing fuel consumption, which is especially important in regional aviation operations which often involve small airports with short runways [4].

The VG on the aircraft wing functions to optimize aerodynamic characteristics by controlling the surrounding air flow. VG creates turbulence or small vortices that help keep the air flow against the surface of the wing, thereby preventing flow separation which can reduce lift [5]. By keeping the flow attached to the wing, the VG increases lift, reduces drag, and in some aircraft, also increases stability during flight maneuvers. This makes it an important element in modern aircraft design, aiming for greater efficiency, performance and maneuverability in a wide range of flight conditions.

The influence of geometric parameters on the VG, which include height, length and shape, greatly influences the aerodynamic characteristics of lift and drag on aircraft wings. This geometric parameter is very dependent on the thickness of the boundary layer (BL) formed on the wing surface. Several types of VG can be designed to sink into the BL, right at the BL or higher than the BL [6]. The installation position and distribution of the wings also have a big influence. These parameters include position relative to the chord, installation angle or distance between pairs. VG is usually installed before the separation point or right at the separation point [7]. Some VG are arranged in pairs co rotating or counter rotating and single VG. Some are arranged in multi rows with inline or staggered arrangement. These arrangement parameters are very dependent on the shape of the airfoil, and the flying conditions of the aircraft. Improper geometry and position can have the effect of increasing the size of the wake region or making flow separation worse.

Agarwal and Kumar [8] conducted research on the NACA 4412 airfoil wing at $Re=10^5$ by varying the position of the VG and at various angle of attack (AoA). In his research, it was found that VG at low AoA provides aerodynamic losses in the form of reduced lift and increased drag. However, at higher AoA, the VG provides significantly increased lift and reduced drag. And the position of the VG is advantageous when it is close to the leading edge of the airfoil [8]. Huang *et al*., [9] studied the phenomenon of shockwave boundary layer interaction (SWBLI) for specific VG on transonic aircraft using the RANS equation model. The VG's row and location in relation to the chord were changed throughout the study process. Consequently, in transonic flights, the VG can lessen wing-tip vortice, spanwise flow, and flow separation behind the SWBLI [9]. Srinath and Sahana [10] conducted research on the VG model using the computational fluid dynamic (CFD) method and experiments with 2 VG models installed at the leading edge and trailing edge. The result is that the pressure above the wing surface increases so that the BL is reenergized and drag decreases [10]. Namura *et al.,* [11] carried out multipoint design optimization on a vane type VG on a swept wing with a common research model (CRM) airfoil using the CFD method to get the right configuration for cruise and critical conditions. As a result, the revised VG design can cut the drag penalty by 50% and stop shock-induced separation from spreading to the wing tips at the CRM wing's Yehudi break [11]. Ito and Yamamoto [12] studied the effects of a corotating blade-type VG on a transonic swept wing using CFD methods and experiments. The result is that the toe out VG configuration is more efficient in the CRM model [12]. Kuwik *et al.,* [13] conducted a study on the design of a VG for the MQ 9 Reaper wing with varying rectangular, delta, round and trapezoidal shapes at an AoA of 0^o with $Re= 5.8 \times 10^6$. The result was that all VG had a negative impact on reducing the lift force by 5.8% to 15%, but the turbulence flow over the wings can be avoided [13].

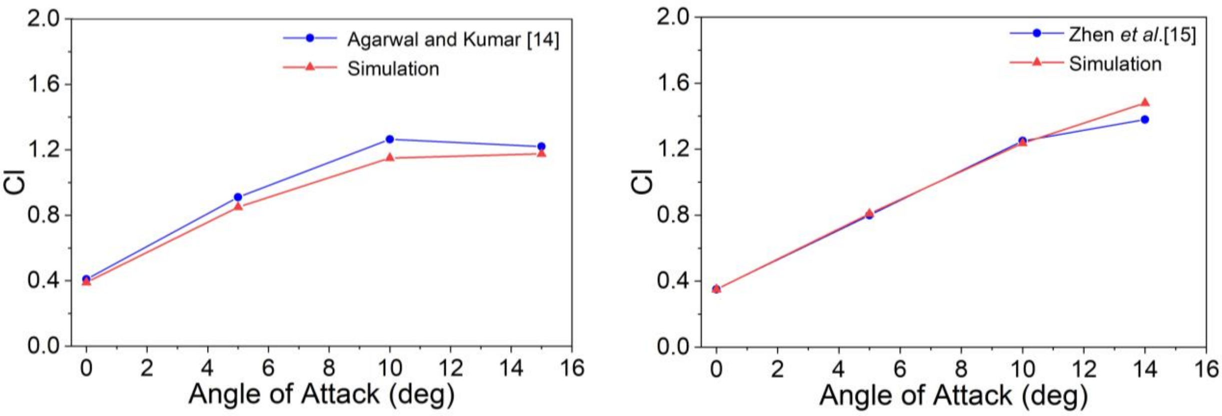
The study of the use of VG on aircraft wings using the Computational Fluid Dynamics (CFD) method is very important considering its crucial role in increasing the aerodynamic efficiency of aircraft. This research is important because it can provide insight into how VG configurations and parameters can be optimized to increase lift, reduce drag, and overall improve the performance of regional aircraft operating at low speeds and high AoA. CFD methods provide powerful predictions for modeling the complex interactions between VG and airflow. The aim of this research is to examine the design of a counter-rotating vane type VG for regional aircraft airfoils in takeoff conditions. The VG configuration obtained is then compared with a wing without a VG at all AoA.

2. Methodology

This research was carried out on small capacity regional aircraft to improve aerodynamic performance, especially in critical take-off and landing conditions on short runways, namely at a speed of 30 m/s and an AoA of 15°. Validation is carried out by comparing research from previous journals to validate the program. Optimization is carried out on geometric parameters and arrangement parameters to see changes in the values of lift, drag and flow behavior. Various VG shapes, heights and lengths are optimized to get optimal results. And various installation positions such as angle of incidence, chord location, pair distance are also studied to obtain an efficient design.

2.1 Validation

Validation is carried out to ensure that the model and simulation settings are correct according to previous research. In order to verify the accuracy of the simulation, it is compared to the findings published in reputable journals that were similar in terms of the objects, phenomena and output taken. All geometric conditions, meshing and simulation settings are equalized to obtain similar results. Simulated lift coefficient (Cl) from the results was compared with Cl from research by Agarwal and Kumar [14], regarding steady simulation of the NACA 4412 airfoil wing with the S-A model at a speed of 20 m/s. Meanwhile, comparison of the results with research by Zhen *et al*., [15], regarding transient simulations on the NACA 4415 airfoil at a very low speed of 3.625 m/s, a graph of the ratio of lift force to AoA is obtained in Figure 1(b).



(a) (b)

Fig. 1. Lift coefficient comparison with Agarwal and Kumar's results (a) and Zhen's results (b)

From the results of these two validations, it can be concluded that the largest error value was obtained in research by Agarwal and Kumar at 9% with an average error of 5.9% and the largest error for research by Zhen *et al*., was 7% with an average error of 1.8%. So, it can be concluded that the simulation model used is valid and can be used for this research simulation.

2.2 Model Geometry

The simulation was carried out on a finite wing with airfoil from the regional aircraft under study. The airfoil chord length is equivalent to the wing MAC length and the wing section width of 1 m. The parameters studied in this research are divided into 2 types, namely geometric parameters and arrangement parameters. The geometry of the VG is shown in Figure 2 and Table 1.

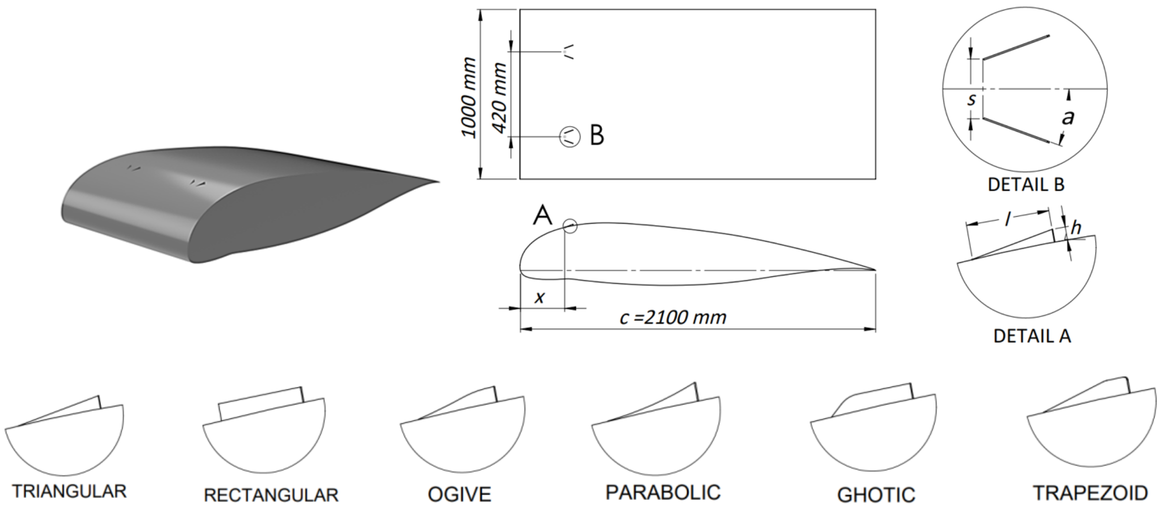


Fig. 2. VG geometry

Table 1
Vortex generator geometry

Geometry	Value
BL height (δ)	0.4%*c*
VG height (*h*)	0.75δ; 1δ; 1.25δ; 1.5δ; 1.75δ
VG shape	Triangular; Rectangular; Ogive; Parabolic; Gothic; Trapezoid
VG length (*l*)	2*h*; 3.5*h*; 5*h*; 6.5*h*; 8*h*
Leading edge distance (*x*)	5%*c*; 7.5%*c*; 10%*c*; 12.5%*c*; 13.8%*c*; 15%*c*
Angle of incidence (*a*)	13°; 16.5°; 20°; 23.5°; 27°
Pair spacing (*s*)	3δ; 4δ; 5δ; 6δ; 7δ
Spanwise spacing	Fixed at 20%*c*

2.3 Fluid Domain

Determining the size of the fluid domain is a critical step that affects the accuracy of the calculation. In the case of external flow, the fluid domain must be made sufficient to represent the fluid that is affected by the air flow on the wing but can still be handled by computer capabilities. The Body of Influence domain is used to detail the behavior of the airflow around the wing and the wake turbulence area behind the wing. The size of the fluid domain for this case is shown in Figure 3.

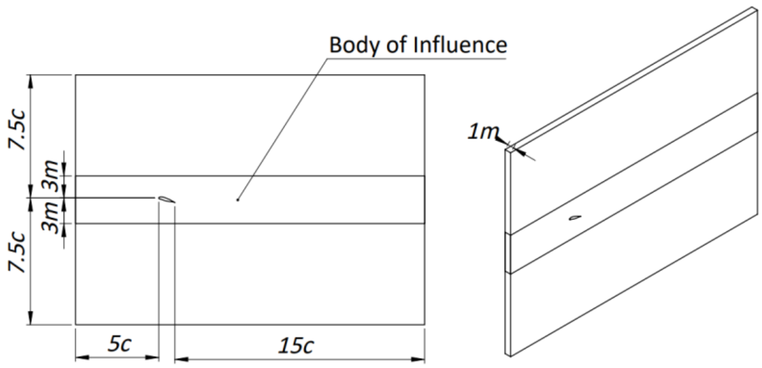


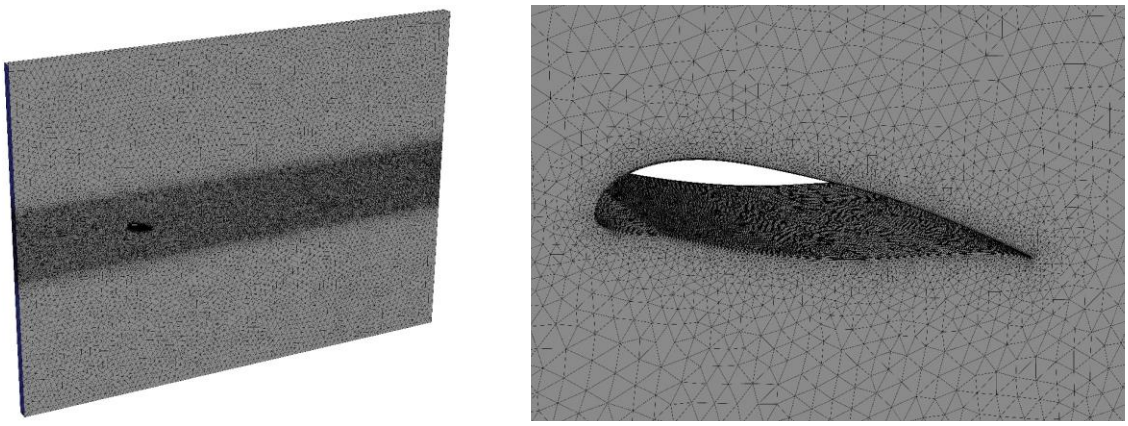
Fig. 3. Fluid domain

2.4 Meshing

The meshing process is an important process that involves the formation of discrete elements that cover the fluid flow area in the VG. Mesh quality and meshing independence affect the accuracy of the simulation [16]. In this model, meshing uses the tetrahedron model because it is able to handle complex geometry and has better representation when reviewing the BL. The meshing settings and quality are shown in Table 2. And the geometry mesh is shown in Figure 4.

Table 2
Mesh setting

Item	Value
General element size	500 mm
Body of influence element size	100 mm
Wing element size	10 mm
VG element size	0.5 mm
Inflation	Maximum thickness 10mm, 4 layer
Skewness	0.88933
Element	3098055



(a) (b)

Fig. 4. Geometry mesh

Before simulation, mesh is tested using a grid independence test to determine the sensitivity of the results to the number and size of mesh elements. When the simulation results do not change much when the number of mesh elements is increased, it means the mesh is valid. The grid independence test graph is shown in Figure 5.

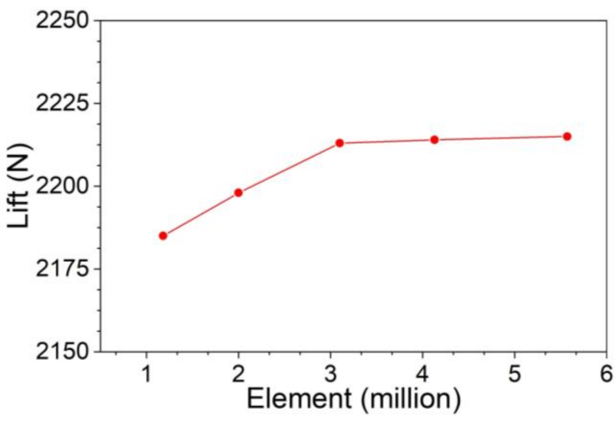


Fig. 5. Grid independence test

2.5 Simulation Set Up

Simulations were carried out with ANSYS Fluent transient using Spalart Allmaras (S-A) model. The S-A model is used because of its wide application in aerodynamic analysis, which has been proven empirically in many cases [17]. This model is also quite good at modelling flow in the BL, which is an important aspect in VG optimization. The SIMPLE scheme is used because it has strong performance, uses relatively low memory and stable convergence, making it a suitable solving engine for a wide range of CFD problems [18]. SIMPLE is also specifically designed to handle pressure-velocity coupling problems well [19]. This is beneficial when there is a complex interaction between the pressure distribution around the wing and the airflow pattern. Second order upwind in spatial discretization is used because it provides better numerical diffusion reduction and maintains the gradient of the flow variable with second order accuracy, which means the error will be smaller than first order [20]. Residual 10^{-3} is used because judging from the force and lift values taken, the data has not changed significantly in iterations below 10^{-3}. The 1.5 second time is determined from the residence time of the fluid in the enclosure from inlet to outlet, also look at the lift and drag graph plot which shows stability at that time. Simulations set up are shown in Table 3.

Table 3
Simulation set up

Item	Setting
Time	Transient
Model	Spalart Allmaras
Scheme	SIMPLE
Pressure	Second order
Momentum	Second order upwind
Turbulent kinetic energy	Second order upwind
Turbulent dissipation rate	Second order upwind
Initialization	Standard
Residual	10^{-3}
Time step size	0.005
Number of time step	300

3. Results

3.1 Plain Wing Test

This stage aims to understand the behavior of regional aircraft wings in stall conditions without a VG during the take-off/landing phase. Plain wing test data is needed to find the airfoil stall angle and comparative data for the lift and drag at all AoA when the VG is installed. Data from plain wing testing results on the airfoil being studied are shown in Table 4, Figure 6 and Figure 7.

Table 4
Plain wing test

AoA	Lift (N)	Drag (N)	L/D
12°	2023.24	66.25	30.54
13°	2100.00	75.02	27.99
14°	2157.18	84.96	25.39
15°	2213.73	94.33	23.47
16°	2263.17	103.43	21.88
17°	2320.90	115.42	20.11
18°	2365.11	128.03	18.47

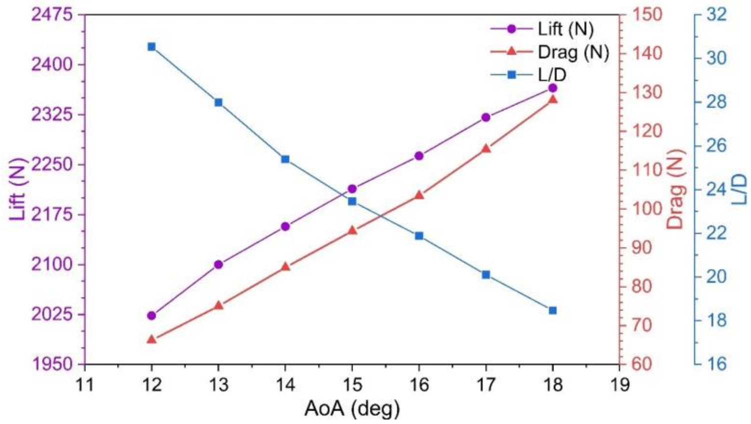


Fig. 6. Plain wing test

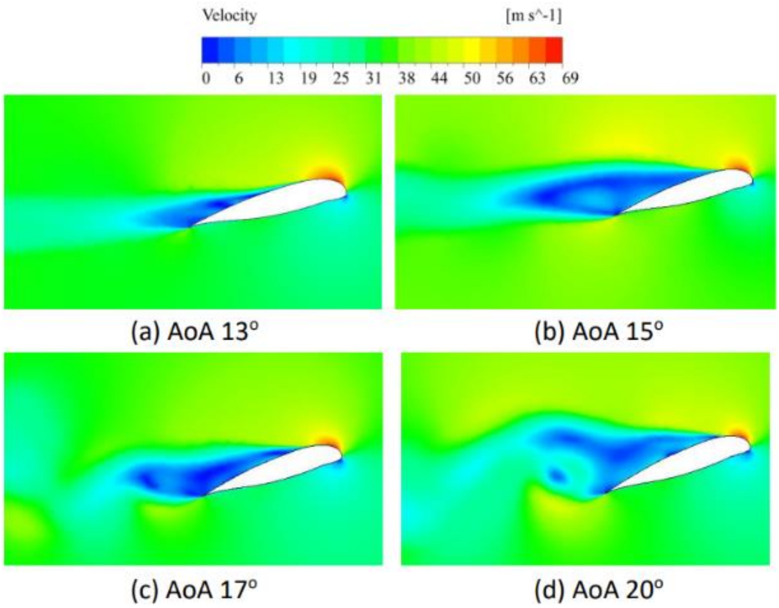


Fig. 7. Velocity contour of plain wing test

3.2 Effect of VG Height

The height of the VG varies depending on the thickness of the BL at the separation point. Based on research from Namura *et al.,* [11], the BL thickness is around 0.4% of the chord length. The height of the VG varies from $0.75\delta \leq h/\delta \leq 2\delta$. Other parameters are kept constant with a triangular shape, leading edge distance 12.5%*c*, length 5*h*, angle of incidence 20° and pair spacing 5δ. A comparison of the results of each variation can be seen in Table 5, Figure 8 and Figure 9.

Table 5
Effect of VG height to lift, drag and L/D

Height (mm)	Lift (N)	Drag (N)	L/D
Plain Wing	2213.73	94.33	23.47
6.3	2196.37	96.64	22.73
7.5	2188.66	97.04	22.55
8.4	2190.93	101.84	21.51
10.5	2191.00	108.39	20.21
12.6	2184.01	115.76	18.87
14.7	2190.86	124.15	17.65

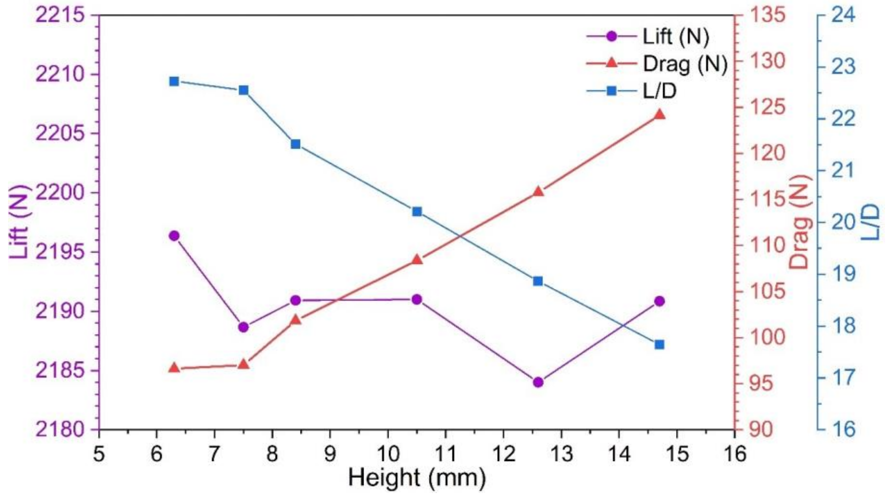


Fig. 8. Effect of VG height to lift, drag and L/D

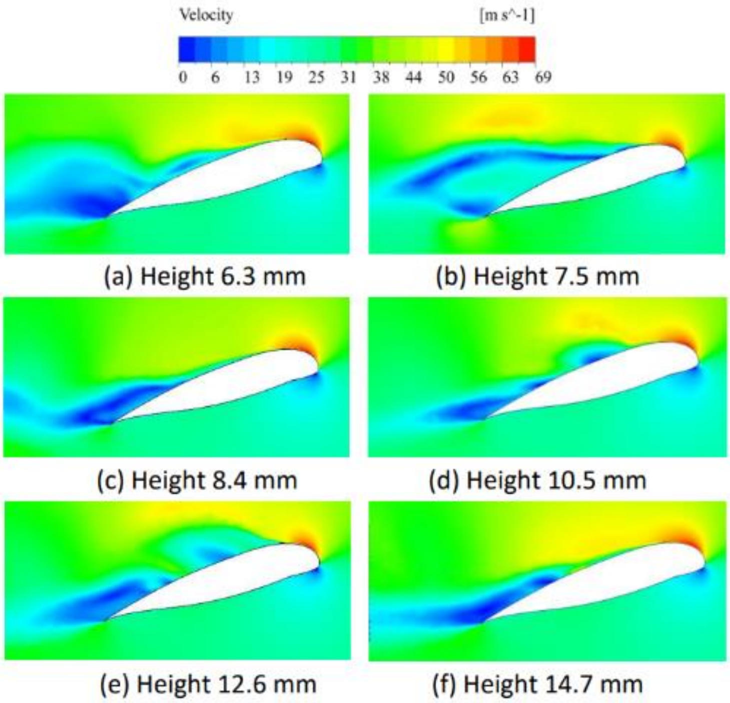


Fig. 9. Velocity contour of height variation

Based on these results, the optimal VG height is produced at a height of 0.75δ. In this geometry, flow separation can begin to be reduced, and the wake region becomes smaller. However, the higher the VG, the more detrimental the effect on lift and drag and the flow becomes very chaotic.

3.3 Effect of VG Shape

The shapes of VG vary widely and each has its own characteristics. In this research, various forms of VG are simulated for regional aircraft take-off/landing conditions. These shapes include triangular, rectangular, ogive, parabolic, gothic and trapezoid. Other parameters are kept constant with height 0.75δ, leading edge distance 12.5%c, length 5h, angle of incidence 20^o and pair spacing 5δ. A comparison of the results of each variation can be seen in Table 6, Figure 10 and Figure 11.

Table 6
Effect of VG shape to lift, drag and L/D

Shape	Lift (N)	Drag (N)	L/D
Plain	2213.73	94.33	23.47
Triangular	2196.37	96.64	22.73
Rectangular	2190.00	103.20	21.22
Ogive	2197.85	96.40	22.80
Parabolic	2191.87	93.80	23.37
Gothic	2191.50	101.25	21.64
Trapezoid	2203.90	100.25	21.98

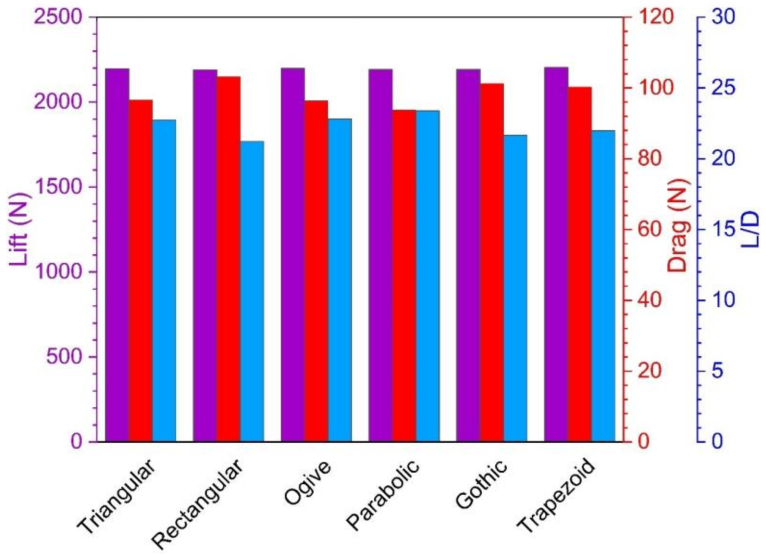


Fig. 10. Effect of VG shape to lift, drag and L/D

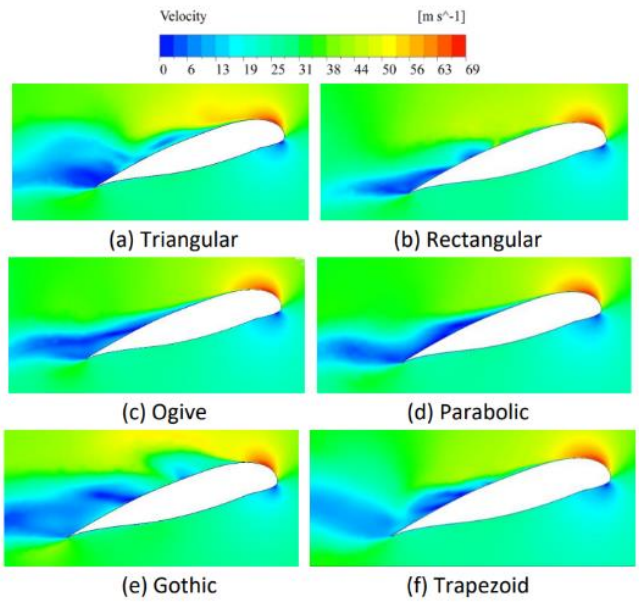


Fig. 11. Velocity contour of shape variation

Based on these results, the optimal VG shape is an ogive shape. This shape provides a better lift drag value than other shapes but has almost the same characteristics as the triangular shape. Meanwhile, the rectangular shape produces a significant decrease in lift and increase in drag and gothic shape provide the most chaotic flow patterns.

3.4 Effect of VG Angle of Incidence

Because this type of VG is counter rotating in pairs, the angle configuration that can be used is toe in and toe out. The toe in configuration is used with variations in the angle of incidence of around 20°, namely 13° ≤ *a* ≤ 27°. Other parameters are kept constant with shape ogive, height 0.75*δ*, leading edge distance 12.5%*c*, length 5*h*, and pair spacing 5*δ*. A comparison of the results of each variation can be seen in Table 7, Figure 12 and Figure 13.

Table 7
Effect of VG angle of incidence to lift, drag and L/D

Angle of incidence	Lift (N)	Drag (N)	L/D
Plain	2213.73	94.33	23.47
13°	2202.00	93.92	23.45
16.5°	2208.90	99.98	22.09
20°	2197.85	96.40	22.80
23.5°	2190.65	99.25	22.07
27°	2195.42	100.50	21.84

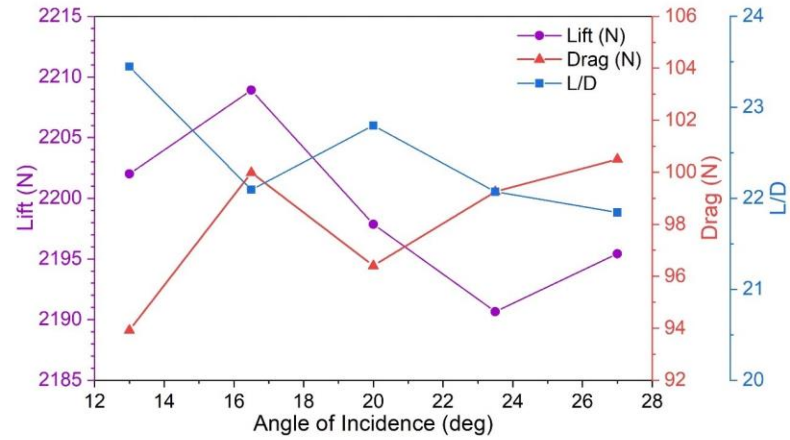


Fig. 12. Effect of VG angle of incidence to lift, drag and L/D

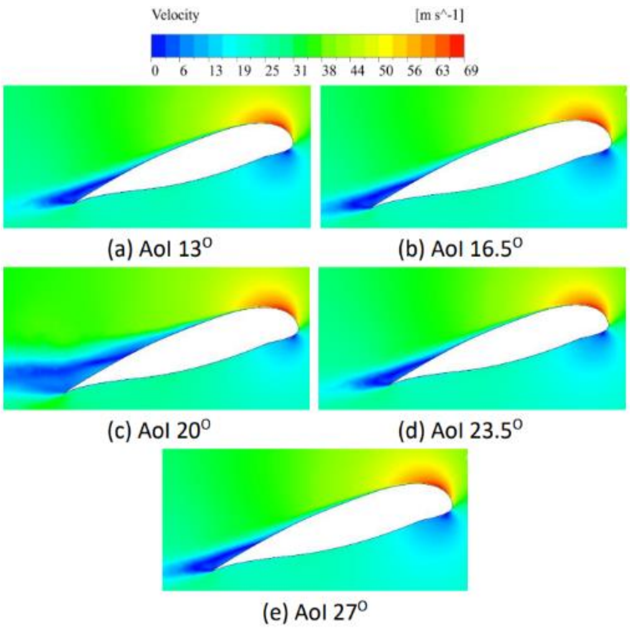


Fig. 13. Velocity contour of angle of incidence variation

Based on the results, the optimum VG is obtained with an angle configuration of 13°. At this angle, increased lift and reduced drag can be achieved compared to the previous configuration and the flow over the wing becomes better with a smaller wake region. In this variation, all angle of incidence values provide a fairly good flow pattern.

3.5 Effect of VG Length

The length of the VG is designed to be sufficient to be able to produce a stable vortex and re-energize the air flow around the wing. The longer it is, the stronger the vortex will be, but the resulting drag will be greater. The length of the simulated VG varies from $2\delta \le l \le 8\delta$. Other parameters are kept constant with shape ogive, height 0.75δ, leading edge distance 12.5%c, angle of incidence 13°, and pair spacing 5δ. A comparison of the results of each variation can be seen in Table 8, Figure 14 and Figure 15.

Table 8
Effect of VG length to lift, drag and L/D

Length (mm)	Lift (N)	Drag (N)	L/D
Plain	2213.73	94.33	23.47
12.6	2207.00	97.00	22.75
22.05	2202.65	95.20	23.14
31.5	2202.00	93.92	23.45
40.95	2207.80	96.16	22.96
50.4	2202.00	97.40	22.61

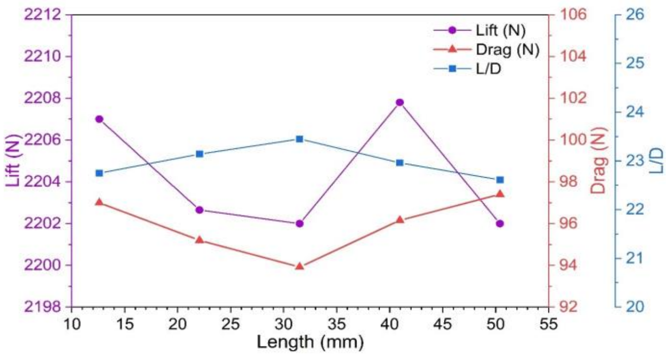


Fig. 14. Effect of VG length to lift, drag and L/D

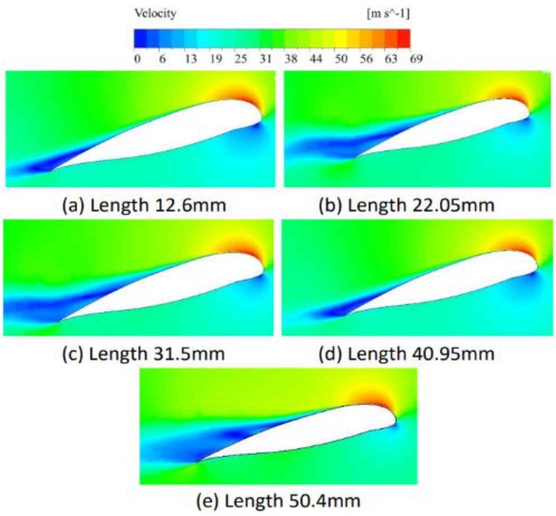


Fig. 15. Velocity contour of length variation

Based on the results, the length of the VG that provides the best effect is 6.5h. In this configuration, the separation flow can be reduced. Even though the L/D ratio is smaller, the lift formed is larger and flow separation can be reduced significantly. The 12.6 mm length also provides quite good lift and drag values and a flow pattern with a small wake region.

3.6 Effect of Pair Distance

In counter rotating VG, the distance between pairs is an important aspect that determines the flow characteristics of the counter rotating vortex. The counter rotating flow at a closer distance between pairs causes the vortex to become stronger due to the interaction between two opposing vortices. The distance between pairs varies from $2\delta \leq s \leq 8\delta$. Other parameters are kept constant with shape ogive, height 0.75δ leading edge distance 12.5%c, angle of incidence 13^{o}, and length 6.5h. A comparison of the results of each variation can be seen in Table 9, Figure 16 and Figure 17.

Table 9
Effect of VG pair distance to lift, drag and L/D

Pair distance (mm)	Lift (N)	Drag (N)	L/D
Plain	2213.73	94.33	23.47
25.2	2200.15	97.01	22.68
33.6	2205.00	96.26	22.91
42	2207.80	96.16	22.96
50.4	2200.00	95.00	23.16
58.8	2204.57	97.17	22.69

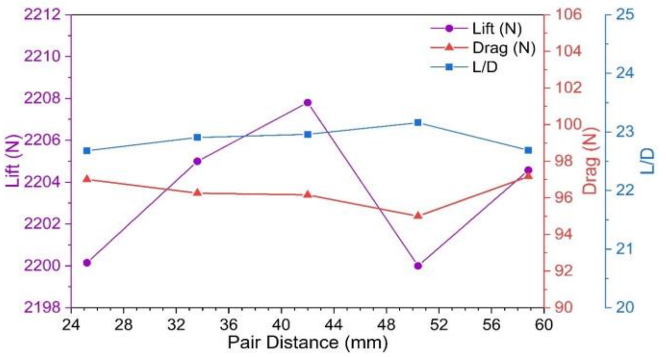


Fig. 16. Effect of VG pair distance to lift, drag and L/D

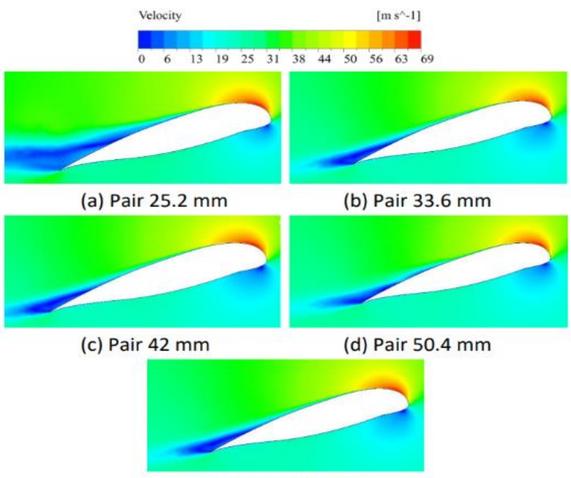


Fig. 17. Velocity contour of pair distance variation

Based on the results, the optimum distance between pairs is 5δ. Too close distance will have a bad effect on lift and a more turbulent flow because the interaction of two vortices that are too close can break the formation of both vortex.

3.7 Effect of Leading Edge Distance

The placement of the VG also depends on the flow separation point at a certain AoA. Usually placed close to the separation point to reattach the separated flow. The distance of the VG from the leading edge varied from 5%$c \le x \le$ 15%c. Other parameters are kept constant with shape ogive, height 0.75δ, pair distance 5δ, angle of incidence 13o, and length 6.5h. A comparison of the results of each variation can be seen in Table 10, Figure 18 and Figure 19.

Table 10
Effect of leading edge distance to lift, drag and L/D

Leading edge distance	Lift (N)	Drag (N)	L/D
Plain	2213.73	94.33	23.47
5%	2193.40	98.85	22.19
7.5%	2203.37	99.90	22.06
10%	2198.20	96.00	22.90
11%	2199.60	95.50	23.03
12.5%	2202.00	97.18	22.66
13.1%	2209.54	98.00	22.55
13.8%	2214.18	98.65	22.44
14.4%	2203.60	96.50	22.84
15%	2204.70	94.82	23.25

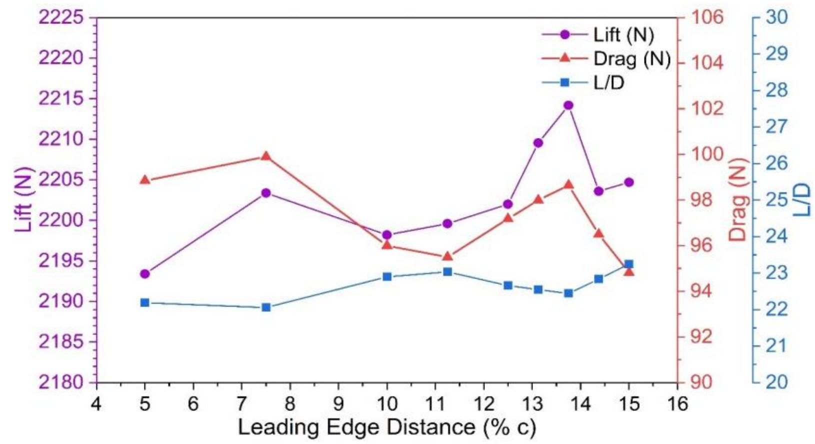


Fig. 18. Effect of leading edge distance to lift, drag and L/D

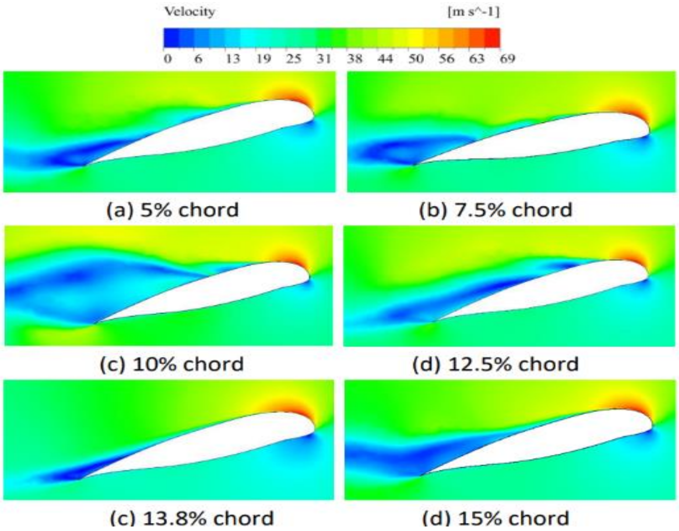


Fig. 19. Velocity contour of leading edge distance variation

Based on the results, the optimal position to get a better lift is 13.8%c. The location of the VG must be chosen correctly because if it is not suitable for the separation location it will actually worsen the flow separation and increase the wake region area. At this point, there is an increase in lift but the L/D ratio is lower than other positions. In this position, flow separation can be completely avoided, and the airfoil can hold up to a higher AoA. The formation of the vortex with the final configuration can be seen in Figure 20.

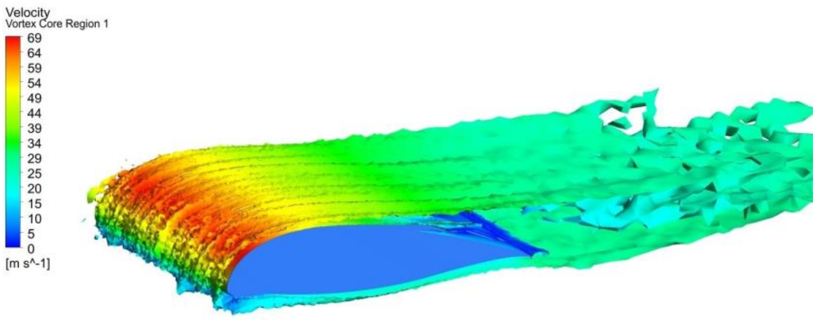


Fig. 20. Vortex contours in final configuration

The best final VG design is with an ogive shape configuration, height 0.75δ, pair distance 5δ, angle of incidence 13^{o}, length 6.5h and located at 13.8% chord. A comparison of the lift and drag values on the wing before and after the VG was installed is shown in Figure 21.

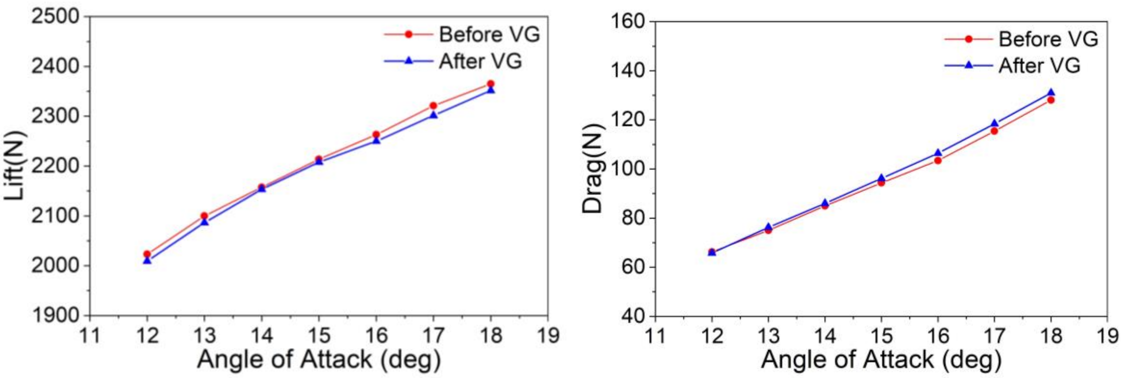


Fig. 21. Comparison of wings before and after VG installed

Based on the comparative data, the counter rotating VG configuration does not have a good effect on increasing lift and reducing drag. Although the air flow around the wing has shown that the flow separation problem can be corrected, it does not always have a good effect on lift and drag. From testing, the final configuration also had negative effects on lift and drag on all AoA.

4. Conclusion

Conclusions from research on counter-rotating vane type wing vortex generators installed on regional aircraft wings using the CFD method is that the use of the VG in all configurations regarding the parameters of shape, height, length, angle of incidence, pair distance and leading edge distance tested does not have the effect of increasing lift or reducing drag. However, the VG successfully overcomes the flow separation problem, which is a positive achievement in this context. The influence of the geometric parameters of the vortex generator is as follows

i. It is better to set the height of the VG submerged in the boundary layer. The greater the height of the VG, the smaller the L/D ratio and the more chaotic the flow becomes.
ii. It is better to use an ogive or triangular VG shape because the lift and drag results are better. Meanwhile, rectangular shapes have the worst effect on aerodynamic characteristics.
iii. The angle of incidence is better installed at a relatively small angle, namely 13^{o}. The larger the angle, the worse the effect on L/D.
iv. The length of the VG is better to use a length of 6.5h because the lift is greater, and flow separation can be reduced compared to other sizes.
v. The distance between pairs is better at 5δ. Too close a distance has a bad effect on the lift and the flow becomes more turbulent because the interaction of the two vortices can break the two vortices apart.
vi. It is very important to pay attention to the distance to the leading edge because the wrong position will actually damage the flow by increasing the separation that occurs.

The test results also show that the use of a VG in the tested configuration produces adverse effects at all attack angles analyzed. Therefore, further studies are needed to evaluate the potential for using VG with other configurations, such as using a single VG or a co-rotating arrangement. While VG can overcome the separate flow problem, its impact on lift and drag needs to be further evaluated, and configuration variations need to be considered to achieve better results in aircraft wing design.

Acknowledgment
This research was made possible thanks to financial support provided by the National Research and Innovation Agency (BRIN). This funding has been instrumental in the successful completion of this research.

References

[1] Babikian, Raffi, Stephen P. Lukachko, and Ian A. Waitz. "The historical fuel efficiency characteristics of regional aircraft from technological, operational, and cost perspectives." *Journal of Air Transport Management* 8, no. 6 (2002): 389-400. https://doi.org/10.1016/S0969-6997(02)00020-0
[2] Merryisha, Samuel, and Parvathy Rajendran. "Experimental and CFD analysis of surface modifiers on aircraft wing: A review." *CFD Letters* 11, no. 10 (2019): 46-56.

[3] Forster, K. J., and T. R. White. "Numerical investigation into vortex generators on heavily cambered wings." *AIAA journal* 52, no. 5 (2014): 1059-1071. https://doi.org/10.2514/1.J052529
[4] J. Meyer, P. Okfen, and C. Bil, "Optimisation of Vortex Generators for Stall Speed Reduction," 2021.
[5] Reichert, B. A., and B. J. Wendt. "Improving curved subsonic diffuser performance with vortex generators." *AIAA journal* 34, no. 1 (1996): 65-72. https://doi.org/10.2514/3.13022
[6] Lee, Sang, and Eric Loth. "Supersonic boundary-layer interactions with various micro-vortex generator geometries." *The Aeronautical Journal* 113, no. 1149 (2009): 683-697. https://doi.org/10.1017/S0001924000003353
[7] Koike, Masaru, Tsunehisa Nagayoshi, and Naoki Hamamoto. "Research on aerodynamic drag reduction by vortex generators." *Mitsubishi motors technical review* 16 (2004): 11-16.
[8] Agarwal, Shubham, and Priyank Kumar. "Numerical investigation of flow field and effect of varying vortex generator location on wing performance." *American Journal of Fluid Dynamics* 6, no. 1 (2016): 11-19.
[9] Huang, Jingbo, Song Fu, Zhixiang Xiao, and Miao Zhang. "Study of separation control of vortex generators on transonic wings." *Journal of Fluid Science and Technology* 6, no. 1 (2011): 85-97. https://doi.org/10.1299/jfst.6.85
[10] R. Srinath and D. Sahana, "Energization of Boundary Layer Over Wing Surface By Vortex Generators," 2017.
[11] Namura, Nobuo, Koji Shimoyama, Shigeru Obayashi, Yasushi Ito, Shunsuke Koike, and Kazuyuki Nakakita. "Multipoint design optimization of vortex generators on transonic swept wings." *Journal of Aircraft* 56, no. 4 (2019): 1291-1302. https://doi.org/10.2514/1.C035148
[12] Ito, Yasushi, Kazuomi Yamamoto, Kazuhiro Kusunose, Shunsuke Koike, Kazuyuki Nakakita, Mitsuhiro Murayama, and Kentaro Tanaka. "Effect of vortex generators on transonic swept wings." *Journal of Aircraft* 53, no. 6 (2016): 1890-1904. https://doi.org/10.2514/1.C033737
[13] Kuwik, Brett, Christopher Tabacjar, and Seong-jin Lee. "CFD investigation of vortex generator additions to the general atomics MQ-9 reaper." In *2018 AIAA Aerospace Sciences Meeting*, p. 1009. 2018. https://doi.org/10.2514/6.2018-1009
[14] Agarwal, Shubham, and Priyank Kumar. "Numerical investigation of flow field and effect of varying vortex generator location on wing performance." *American Journal of Fluid Dynamics* 6, no. 1 (2016): 11-19.
[15] Zhen, Tan Kar, Muhammed Zubair, and Kamarul Arifin Ahmad. "Experimental and numerical investigation of the effects of passive vortex generators on Aludra UAV performance." *Chinese Journal of Aeronautics* 24, no. 5 (2011): 577-583. https://doi.org/10.1016/S1000-9361(11)60067-8
[16] Thornburg, Hugh. "Overview of the PETTT Workshop on Mesh Quality/Resolution, Practice, Current Research, and Future Directions." In *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, p. 606. 2012. https://doi.org/10.2514/6.2012-606
[17] Bueno-Orovio, Alfonso, Carlos Castro, Francisco Palacios, and Enrique Zuazua. "Continuous adjoint approach for the Spalart-Allmaras model in aerodynamic optimization." *AIAA journal* 50, no. 3 (2012): 631-646. https://doi.org/10.2514/1.J051307
[18] Acharya, S., B. R. Baliga, Kailash Karki, J. Y. Murthy, C. Prakash, and Surya Pratap Vanka. "Pressure-based finite-volume methods in computational fluid dynamics." (2007): 407-424. https://doi.org/10.1115/1.2716419
[19] Latimer, B. R., and A. Pollard. "Comparison of pressure-velocity coupling solution algorithms." *Numerical Heat Transfer* 8, no. 6 (1985): 635-652. https://doi.org/10.1080/01495728508961876
[20] De Rango, S., and D. W. Zingg. "Higher-order spatial discretization for turbulent aerodynamic computations." *AIAA journal* 39, no. 7 (2001): 1296-1304. https://doi.org/10.2514/2.1472