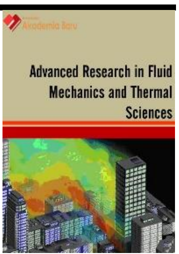
Penerbit
Akademia Baru

Journal of Advanced Research in Fluid
Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Comparison Study Between Schrenk's Approximation Method and Computational Fluid Dynamics of Aerodynamic Loading on UAV NACA 4415 Wing

Open Access

Nur Nabillah Mohd Kamal[1], Adi Azriff Basri[1,*], Ernnie Illyani Basri[1], Ida Suriana Basri[2], Mohd Firdaus Abas[1]

[1] Department of Aerospace, Faculty of Engineering, University Putra Malaysia, 43400 Serdang, Selangor, Malaysia
[2] English Unit Student Development Section, UNIKL Malaysia France Institute, 43650 Bandar Baru Bangi, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 7 October 2019
Received in revised form 1 November 2019
Accepted 3 November 2019
Available online 28 December 2019

ABSTRACT

In early stage of wing structural design, the most important stage of structural analysis is to estimate the aerodynamics loading acting on wing. Years ago, several methods were used to estimate the aerodynamics distribution along span wise and were invented by few scientists. Each method takes different parameters into consideration. Yet, there is no proof whether these theoretical methods are able to provide an acceptable accuracy in estimating wing lift distribution. Schrenk method is commonly used among engineers in the early phase of an aircraft designing process based on its ability to provide fast result. Due to the advancement in technology, a computational fluid dynamics (CFD) has been created in order to get the high-level accuracy in aerodynamics prediction, where it uses Navier-Stokes equation to solve the simulation. NACA 4415 on Aludra MK1 UAV has been chosen to represent an aerodynamics analysis along with the wing span. This study focuses in verifying the Schrenk Approximation method using the computerize data-based analysis (CFD). The goal of this study is to determine the lift distribution along the NACA 4415 wing span and to acquire the percentage errors between Schrenk Approximation method and CFD method. From both studies, a significant aerodynamics lifting is obtained where the percentage difference between both methods are 3% for lift force and 13% for lift coefficient. Hence, Schrenk approximation method can be used for further application of aerodynamics lifting of UAV wing especially for structural analysis since the result proven within the acceptable and reasonable limit with CFD method.

Keywords:
Schrenk approximation method; Computational Fluid Dynamics; NACA 4415; UAV wing; aerodynamic lifting

1. Introduction

Wing is the main part of aircraft that generates lift at most. It has a special shape, which called an airfoil. An airfoil shape helps in reducing pressure on top of the wing surface. Thus, it generates lift force when air goes faster towards it. As known, designing wing would be the most important part in

---

[*] *Corresponding author.*
E-mail address: adiazriff@upm.edu.my (Adi Azriff Basri)

the initial phase of designing an aircraft because it decides whether the lift force is powerful enough to fly the aircraft or not. This process has been implemented years ago since the era of commercial aircraft.

In this work, a comparison between two methods which best suit to estimate aerodynamic loads on Unmanned Aerial Vehicle (UAV) wings is proposed. Both approaches, Schrenk method and Computational Fluid Dynamics (CFD) are suitable for early estimation of aircraft loads due to its simplicity and reasonable accuracy.

Schrenk method was proposed by Dr. Ing Oster Schrenk in 1940 [1], as the theoretical approach, it takes the average lift per unit span between planform lift and elliptical lift [2]. The Schrenk method is based on the fact that the distribution of lift along with an unswept wing does not differ significantly. Schrenk [1] highlighted the criteria required for performing elliptical lift distribution are: upswept wing and zero lift generated at the tip and maximum lift generated at the center lines along the wing span. This research works on Schrenk approximation method received less attention due to the growing application of CFD. Till date, only few applications of Schrenk approximation method available in UAV field, especially by Hutagalung, Latif and Israr, Putra *et al.,* and Soemeryanto and Rosid [2-4].

CFD is one of the numerical analysis and algorithm of fluid mechanics that being used to analyze and solve the fluid flow problems. In fluid mechanics, the continuum mechanics equation or known as transport equations were taken from the Navier-Stoke equations where these equations explained the physics of fluids movement in the Newtonian mechanics frameworks. Several researches on this method has been carried out by Karansinghdangi and Mathur, Panagiotou *et al.,* and Sahin [5-7].

The goal of this study is to determine the lift distribution along the wingspan by using both methods. Then, the obtained lifting values of both methods will be compared in terms the percentage differences.

This paper consists of five sections, arranged in a sequence begin with introduction in section 1, followed by modelling of UAV in section 2. The third section is methodology, while section 4 exhibits the result of this research together with a discussion. Then, in the final section is conclusion of work.

2. Modelling of UAV

Aludra MK1 was developed to be employed in military sector, for battlefield surveillance and reconnaissance mission. Not only focuses to these two missions, other applications are traffic monitoring, disaster area monitoring, law enforcement etc. This aircraft was programmed either to be autonomous flight with a critical flight plan or remotely controlled from ground, Tactical Ground Control Station (GCS) and able to transmit real-time video imagery to the ground station [8]. Table 1 shows the technical specification of Aludra MK1 and Table 2 provides the wing specifications.

Table 1
Specification of Aludra [website www.ctrm.com.on 5 April 2019]

Specifications	Values
Weight	200kg MTOW, max payload
Airfoil	NACA 4415
UAV Length	4.572 m
Wingspan	5.1257 m
Endurance	3 hours
Max speed	220 km/h or 61.11 m/s

Table 2
Aludra MK1 wing specification

Aludra MK1 – NACA 4415	
Wingspan, b	5.1257 m
Wing surface area, S	3.1899 m2
Chord length, C	0.5886 m
Taper ratio	1
Aspect ratio, AR	8.2363

3. Methodology

3.1 Schrenk Method

This method is a simple approximation method to estimate the lift force distribution on wings. Schrenk methods approximate the force distribution by taking the average of lift per unit span between elliptical lift and planform lift distribution. The mathematical model of this method is as shown in Eq. (1) to (3).

Elliptical Lift,

$$L'_{elliptical} = \frac{4S}{\pi b}\sqrt{1-\left(\frac{2y}{b}\right)^2} \tag{1}$$

Planform Lift,

$$L'_{planform} = \frac{2S}{(1+\lambda)b}\left(1+\frac{2y}{b}(\lambda-1)\right) \tag{2}$$

Schrenk Lift,

$$L'_{Schrenk} = \frac{L'_{elliptical}+L'_{planform}}{2} \tag{3}$$

The lift distribution on the wing is extracted by using Schrenk's method where the load distribution resulted in elliptic shape along of the wingspan. The calculations were computed for the wing loading based on the half wingspan. The data for the airfoil required in completing Schrenk's approximation is referred to Table 2.

The steps of the Schrenk's calculation are further elaborated.

i. Divide the half wingspan into 128 sections, from wing root to wing tip with an interval value of 0.02.
ii. Calculate the elliptical chord distribution.

$$C_{elliptical} = \frac{4S}{\pi b}\sqrt{1-\left(\frac{2y}{b}\right)^2}$$

iii. Calculate the modified planform chord distribution.

$$C_{modified} = C_{original} - \left(C_{original} \times 0.2\right)$$

The presence of fuselage contributes in generating lift for the aircraft. This cause to loss in lift generated by the wing due to the percentage of lift generated by the fuselage. The percentage of lift loss is assumed to be 20% along the wing, taking as the general value among all types of aircraft which the percentage is below than 20% [9].

i. Compute for actual lift coefficient (CC_L) which Schrenk distribution.

$$CC_L = C_{Schrenk} = \frac{1}{2}\left[C_{modified} + C_{elliptical}\right]$$

ii. Determine the lift force from the actual lift coefficient.

$$L_n = \frac{1}{2} \times \rho \times v^2 \times S \times CC_L$$

Hence, the lift distribution along the wing resulted in elliptical shape as depicted in Figure 1.

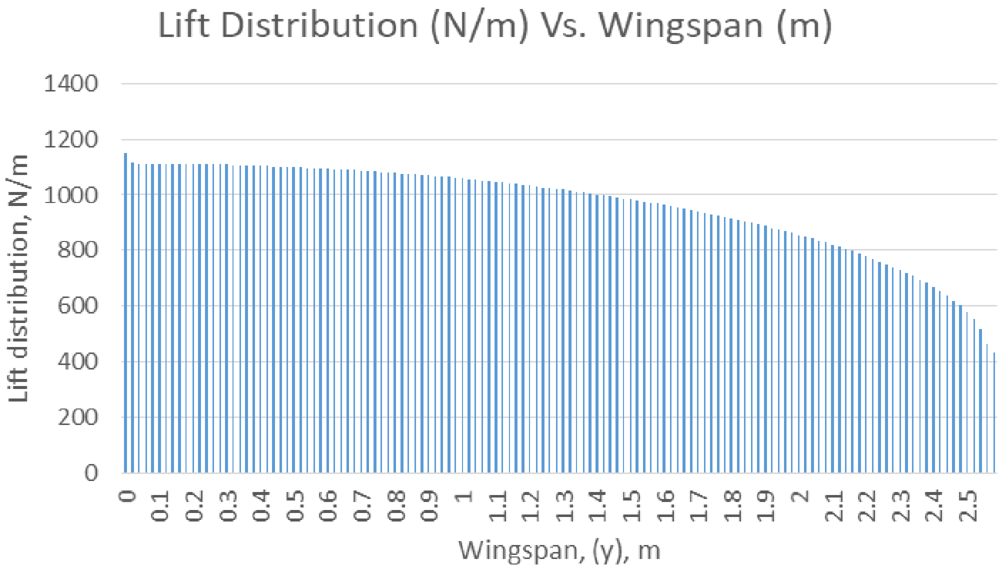


Fig. 1. Lift force distribution along the wingspan based on Schrenk calculation

3.2 Computational Fluid Dynamics (CFD) method

The computational analysis of CFD is performed using ANSYS Fluent software. The whole work conducted in CFD is presented in Figure 2.

The Navier-stokes equation as a common general equation is used to represent the fluid flow behavior. One of the numerical analysis and algorithm used to solve the fluid flow problem, in ensuring a high-level accuracy of predicting the aerodynamic load [10]–[15]. The flow is assumed as incompressible flow and the Navier-Stokes equation is represented as Eq. (4).

$$\rho\left[\frac{\partial u}{\partial t} + (u\nabla)u\right] = -\nabla p + \mu\nabla^2 u + \rho F \tag{4}$$

ρ : Density (kg/m^3)
$\frac{\partial u}{\partial t}$: Time derivative of velocity

p : Pressure (Pa)
F : Force (N)
u : Velocity (m/s)

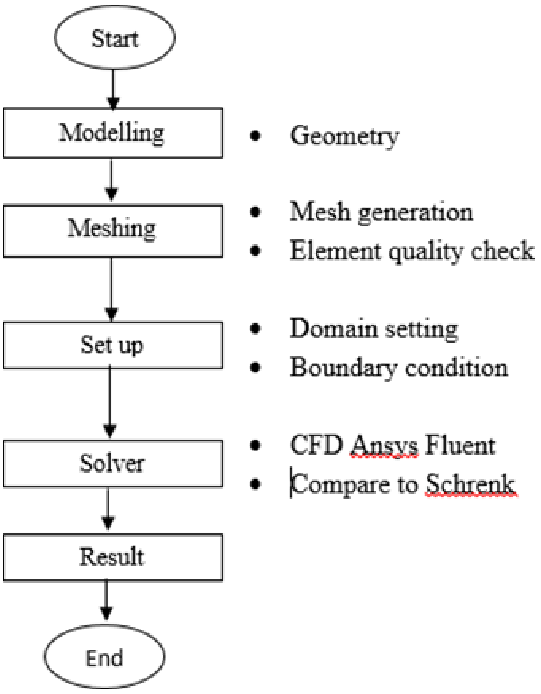


Fig. 2. Workflow of CFD

This study used to simulate the work flow through the airfoil surface and analyze the lift distribution on NACA 4415 Aludra MK1 as per described in Table 2. The wing geometry is developed based on half wing as it corresponds to the other half as shown in Figure 3.

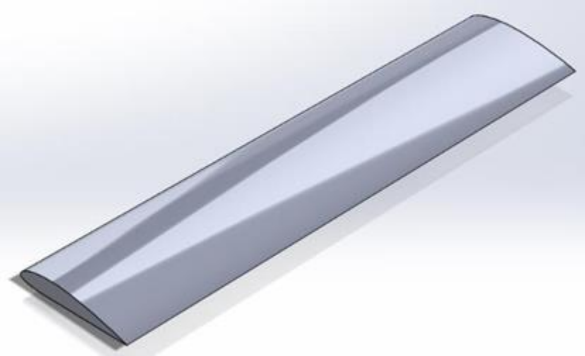


Fig. 3. Half wing geometry of Aludra MK1 wing

In CFD, the structured mesh is conducted by using Pointwise software before exporting it into Fluent solver. Before that, double precision was activated at the beginning of solver to improve accuracy [16]. In ensuring that the flow on boundary layer being captured accordingly, mesh surrounding the surface must be finer compared to the farthest element one, as in Figure 4.

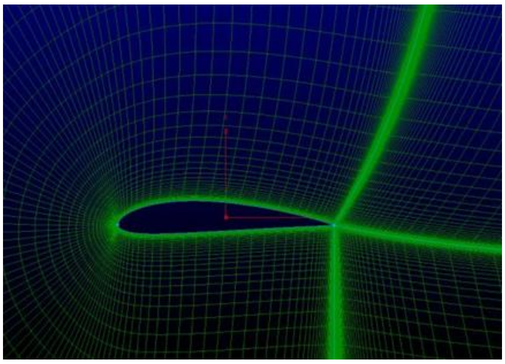


Fig. 4. Mesh surround the wing geometry

3.2.1 Mesh dependency

Mesh dependency has been generated according to lift force in 5 different mesh element, 0.7 million, 1.0 million, 1.3 million, 1.6 million and 2.3 million. This precaution step was done to obtain an accurate result [16]. Figure 5 simulates the mesh dependency check.

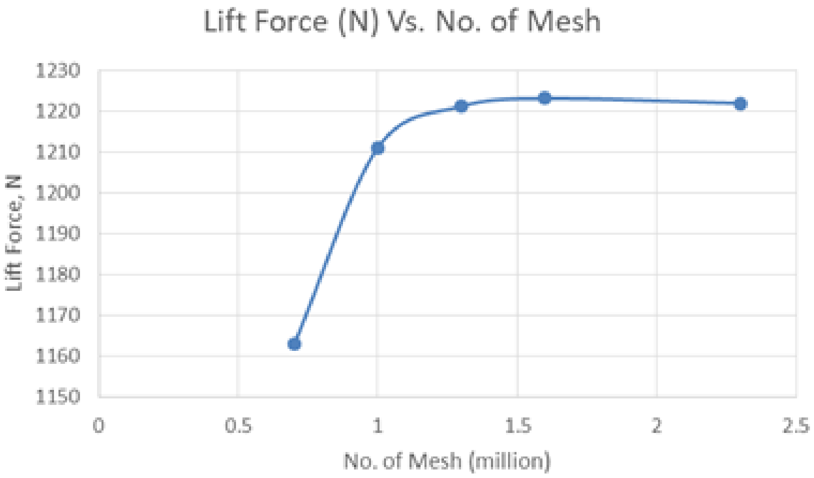


Fig. 5. Mesh dependency graph

From graph, it can be concluded that 1.3m is the optimum mesh to be used in this study. The grid spacing from the wing boundary, first layer of mesh is computed as 0.0114184 mm to achieve a desired y+ which is below than 1 as per calculated in online y^+ calculator.

3.2.2 Boundary condition

In this case, the boundary conditions consist of flow inlet and exit boundary, wall, internal face boundaries plus symmetry and pole boundaries. This step is compulsory which it is used to introduce the solver of the required condition needed. Table 3 shows the boundary conditions set up for the Fluent solver at Standard Temperature and Pressure (STP).

Table 3
Values set up for the boundary conditions

Boundary condition	Values
Velocity inlet	30.55 m/s
Pressure outlet	0 Pascal
Fluid type	Air, ideal gas
Turbulence type	*k-ε*
Residual error	10−6

A pre-calculated Reynolds number is compulsory in defining the fluid flow model for the solver [17]. Thus, the value is 1,231,005. Hence, the turbulence model is required in this simulation.

4. Results and Discussion

The result of lift force distribution along the spanwise computed by CFD and Schrenk method is illustrated in Figure 6.

According to Figure 6, blue line with dash pattern indicated the Schrenk result while red line with solid pattern indicated the CFD result. The result is computed according to each small section on wingspan.

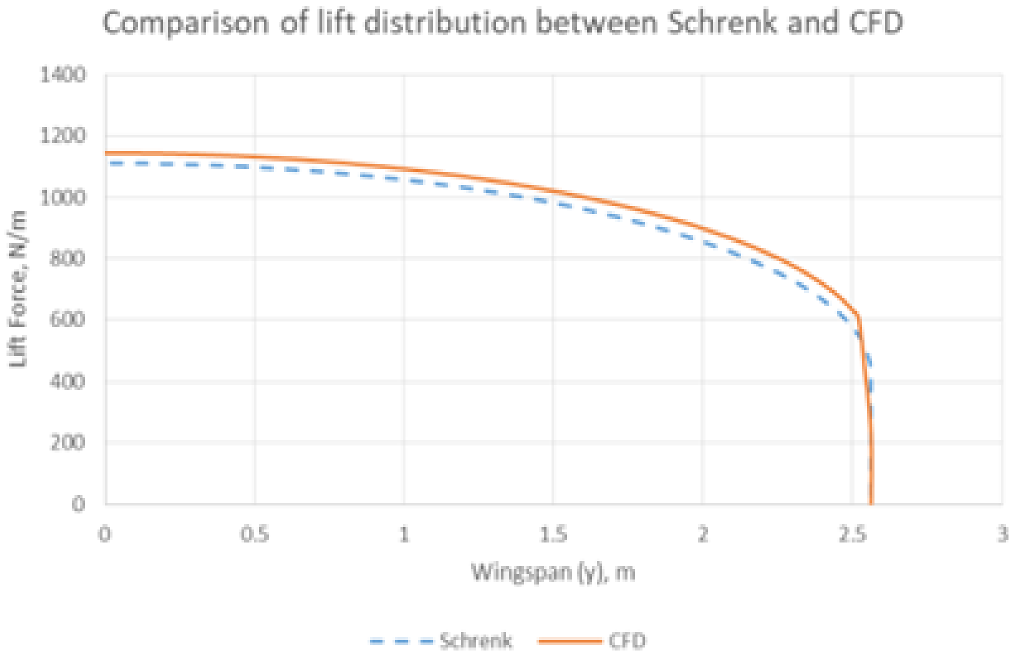


Fig. 6. Lift force distribution along half wingspan of Aludra MK1

The graph trend for both methods is in good agreement, whereby the values computed by using Schrenk method results are slightly less than CFD. Commonly, the CFD result is higher than numerical method used. This is due to the accuracy provided by the CFD simulation in which CFD captured the fluid flow almost exactly on the boundary layer of geometry, thus result is high in accuracy. As can be seen, the resulting lift force was gradually decreased when approaching towards the wing tip. This phenomenon is due to less contribution in generating lift at the wing tip.The higher lift force being generate at the wing is within the area between wing root and middle of the wing, as the lift generated was also being supported by the fuselage.

The aerodynamics data that produced by both methods at 0-degree angle of attack and velocity of 30.55 m/s is shown in Table 4.

As in Table 4, both Schrenk method and CFD showed a good agreement between them as the results between both are insignificantly different. For CFD, the maximum lift force simulated by the software was 1144.9 N/m while for Schrenk method is 1112.6 N/m. Besides, the maximum lift chord coefficient simulated by the CFD is 0.6904 whereas Schrenk method gives the value of 0.6101. As mentioned by the previous paper, the difference between these two methods in terms of lift force was within 15% [2]. Therefore, this paper indicated the percentage difference of lift force between both methods less than 15%, which only 2.9% while for lift coefficient; the percentage difference is about 13%.

Table 4
Aerodynamic result of wing

Properties	Value		Error (%)
	Schrenk	CFD	
Max. lift force	1112.6	1144.9	2.91
Lift coefficient	0.6101	0.6904	13.17

The different values that arise from these two methods are due to some factors such as chamber height is not being taken into account in Schrenk method, uncertain value of fuselage contribution in Schrenk's calculation, which being assumed as 20% based on general contribution by the fuselage. Where, different size and shape of fuselage may lead to significant result in different percentage.

Otherwise, the result obtained from CFD, is in pressure. In reckon the lift simulated by the CFD, values of lift chord coefficient (C_{CL}) is required. Thus, in order to obtain those values, the average value of elliptical chord and chord length are calculated. Lift force is then computed by multiplying the pressure obtained from CFD and C_{CL}. The reason why the actual chord length and not the modified chord as per used in Schrenk, is used in calculating C_{CL} is because in CFD simulation, the geometry used did not include fuselage design which makes no contribution percentage from the fuselage.

Schrenk method does have few constraints regarding its compatibility on wing which some constraints may cause differences between both methods. As stated in literature review, Schrenk method is only applicable on unswept wing, either on straight wing or tapered wing. Thus, the limited application limits Schrenk's ability in estimated lift force on other type of wing. In completing this study, it has been noticed that Schrenk method does not take chamber size of airfoil into account, differ from CFD analysis. CFD captured the fluid flow at the leading edge of the wing where inclination in angle existed at the maximum chamber height.

Besides, several attempts in achieving good agreement between both methods were done by variety the velocity moving towards the wing such as using a maximum cruise velocity of Aludra MK1, randomly chose velocity and average cruise velocity. Yet, only average cruise velocity resulted in good agreement whereas other velocities resulted in high percentage differences between these two methods. This proved that Schrenk method estimated the lift force distribution on wing at average velocity only.

Besides, pressure simulated by the CFD was based on upper and lower part of the wing, differ from Schrenk's where compute lift force at the upper part only. Thus, to compare CFD with Schrenk's, the average value of pressure at each small section of wingspan simulated by the CFD shall be calculated, the maximum and minimum values of pressure are considered.

In the nutshell, the completed work shows a good agreement between CFD and Schrenk. Even though the agreement only shows at average velocity, yet it is sufficient for Schrenk to be used in the initial phase of designing process of an aircraft. By applying Schrenk in estimated the lift force distribution, result in less time consuming and acceptable result in designing process at the early phase.

5. Conclusion

Based on study, there is a significant difference between both approaches, Schrenk and CFD in obtaining lift distribution along the wingspan. As per discuss, the difference may affected by few factors which come from the basic method that being used. As to summarize this, Schrenk does not take chord size into account, uncertain percentage of fuselage contribution was assumed. It is just to approximate the lift distribution based on the average of elliptic and planform distribution over the wing. Whereas, CFD itself provide a result in static pressure over the wing.

Both methods provide a basic result that cannot be compared straightly. Few steps and processes were done before desired values according to selected properties are suitably compared. These processes and steps are done may cause errors in achieving result due to unwanted factors. Nevertheless, this study is completely done with indicated error between both methods are about 3% for lift force and about 13% for lift coefficient. Thus, the study goal is achieved.

Acknowledgement

This work is supported by UPM under GP-IPM grant, 9675000. The authors would like to express their gratitude and sincere appreciation to Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia for the close collaboration in this work.

References

[1] Schrenk, Oskar. "A simple approximation method for obtaining the spanwise lift distribution." *The Aeronautical Journal* 45, no. 370 (1941): 331-336.

[2] Putra, C., Ratih Julistina, Mochammad Agoes Moelyadi, and Taufik Mulyanto. "Comparative Study between Schrenk and CFD Analysis for Predicting Lift Distribution along Wing Span of Glider Aircraft." *Aerospace Science and ‘Technofogy in Indonesia* (2016).

[3] Hutagalung, M. R. A., A. A. Latif, and H. A. Israr. "Structural design of UAV semi-monoque composite wing." *Journal of Transport System Engineering* 1 (2016): 26-34.

[4] Soemaryanto, Arifin Rasyadi, and Nurhayyan Halim Rosid. "Verification of Schrenk method for wing loading analysis of small unmanned aircraft using navierstokes based CFD simulation." *Jurnal Teknologi Dirgantara* 15, no. 2 (2018): 161-166.

[5] Karansinghdangi, Miss Neha Mathur, Seshagiri Rao G.V.R, and Anand Kumar Pathak. "CFD analysis of an aircraft delta wing." *Int. Res. J. Eng. Technol.* 3, no. 12 (2016): 555–560.

[6] Panagiotou, P., P. Kaparos, C. Salpingidou, and K. Yakinthos. "Aerodynamic design of a MALE UAV." *Aerospace Science and Technology* 50 (2016): 127-138.

[7] Pinar Şahin. "Navier-Stokes calculation over swept wings." (Master Diss., Middle East Technical University, 2006).

[8] "ALUDRA MK1 (Alliance Unmanned Developmental Research Aircraft)". Unmanned Systems Technology Sdn Bhd (The ALUDRA MK1). Retrieved 30 August 2015.

[9] T. S. V. Kumar, A. W. Basha, M. Pavithra, and V. Srilekha. "Static & dynamic analysis of a typical aircraft wing structure using Msc Nastran." *Int. J. Res. Aeronaut. Mech. Eng.* 3, no. 8 (2015): 1–12

[10] Basri, Ernnie Illyani, Adi Azriff Basri, Mohd Firdaus Abas, Faizal Mustapha, Mohamed Thariq, Hameed Sultan, and Kamarul Arifin Ahmad. "UAV NACA4415 wing structural performance analysis subjected to external aerodynamic load using Schrenk’s approximation." *Journal of Advanced Research Fluid Mechanics and Thermal Science* 60, no. 2 (2019): 178-190.

[11] Kurukularachchi, Lasantha, Rajeeve Prince, and S. R. Munasinghe. "Stability and control analysis in twin-boom vertical stabilizer Unmanned Aerial Vehicle (UAV)." *International Journal of Scientific and Research Publications* 4, no. 2 (2014): 1-5.

[12] Panagiotou, P., P. Kaparos, and K. Yakinthos. "Winglet design and optimization for a MALE UAV using CFD." *Aerospace Science and Technology* 39 (2014): 190-205.

[13] Dahalan, Md Nizam, Ahmad Fitri Suni, Iskandar Shah Ishak, N. A. R. Nik Mohd, and Shabudin Mat. "Aerodynamic

study of air flow over a curved fin rocket." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 40, no. 1 (2017): 46-58.

[14] Mansor, Shuhaimi, Yasser Nogoud, Shabudin Mat, Md Nizam Dahalan, and Ainullotfi Abdul-latif. "Superaugmented pitching motion of UTM CAMAR UAV using advanced flying handling qualities." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 40, no. 1 (2017): 27-37.

[15] Jamil, M. M., M. I. Adamu, T. R. Ibrahim, and G. A. Hashim. "Numerical study of separation length of flow through rectangular channel with baffle plates." *Journal of Advanced Research Design* 7, no. 1 (2015): 19-33.

[16] Razali, A. A., and A. Sadikin. "CFD simulation study on pressure drop and velocity across single flow microchannel heat sink." *Journal of Advanced Research Design* 8, no. 1 (2015): 12-21.

[17] N. M. Isa, R. A. M. Rushdi, A. Sadikin, S. M. Basharie, M. Universiti, and T. Hussein. "Investigation of the fluid motion with various clearances in biodiesel reactor by using CFD." *Journal of Advanced Research Design* 10, no. 1 (2015): 1–8.