Comparative Study of Wing Lift Distribution Analysis for High Altitude Long Endurance (HALE) Unmanned Aerial Vehicle

Faber Y. Silitonga$^{1,*}$ and M. Agoes Moelyadi$^{1,**}$

$^1$) Department of Aeronautics and Astronautics, Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung, Indonesia.

*faber.yosua@gmail.com  **moelyadimochammad@gmail.com

Abstract. The development of High Altitude Long Endurance (HALE) Unmanned Aerial Vehicle (UAV) has been emerged for both civil and military purposes. Its ability of operating in high altitude with long endurance is important in supporting maritime applications. Preliminary analysis of HALE UAV lift distribution of the wing presented to give decisive consideration for its early development. Ensuring that the generated lift is enough to compensate its own weight. Theroretical approach using Pradtí’s non-linear lifting line theory will be compared with modern numerical approach using Computational Fluid Dynamics (CFD). Results of wing lift distribution calculated from both methods will be compared to study the reliability of it. HALE UAV ITB has high aspect ratio wing and will be analyze at cruise flight condition. The result indicates difference between Non-linear Lifting Line and CFD method.

Keywords: aerodynamic, lift distribution, HALE UAV.

Nomenclatures

- $L$ : Lift
- $L'$ : Lift distribution (per chord)
- $C$ : Chord
- $C_l$ : 2D Lift coefficient
- $C_l$ : 3D Lift coefficient
- $\rho$ : Density
- $P$ : Static pressure
- $V$ : Velocity
- $S$ : Wing area
- $q$ : Dynamic pressure
- $\alpha$ : Angle of attack
- $b$ : Span
- $AR$ : Aspect ratio
- $x, y$ : Coordinates
1. Introduction
Indonesia is a very large country that contains of vast amount of islands, not only a huge archipelago, also rich of natural resources the spread from the land throughout the ocean. Large territory which holds thousands of island, challenges to provide modern technology that able to monitor its large area within certain amount of times. Not only being able to monitor large area with long endurance, human resources availability needs to be taken as consideration. HALE UAV ITB (Institut Teknologi Bandung) development is to fulfill the need of reliable autonomous vehicle in order to monitor Indonesian country border and natural resources, that can also be controlled by remote control and telecommand from ground station. ITB has three years plan to develop HALE UAV that divided into three stages. The first stage produced a 12 m wingspan prototype with four boom configuration with four electric propulsion system for about 20000 ft altitude, as shown in figure 1. First prototype had some problems with side wind effect that cause heavy bending moment, damaging the wing. The second stage, which is currently ongoing project, will have a 16 m wingspan that able to operate at 20000 ft altitude. The second prototype, showed in figure 2, will be using single boom with dual electric motor, which is obviously provides smaller weight than four boom configuration. Larger wingspan means larger aspect ratio, which expected to be producing spanwise lift distribution more similar with rectangular lift distribution approach due to relatively smaller induced drag produced. If the hypothesis is satisfied thus, winglet addition can be ignored that will reducing wing bending moment caused by side wing effect.

Spanwise lift distribution prediction can be used to predict the structural load, which is important in designing the structural initial sizing and configuration of the wing. Prandtl’s Non-linear Lifting Line Theory, is choosen rather than Prandtl’s Classic Lifting Line Theory due to is ability to predict the non-linear $C_l$ vs alpha curve that usually happened at high angle of attack thus, expected to produce more accurate result with relatively less time consumed, to CFD simulation. On the other hand, CFD requires longer amount of time but calculate more accurate lift distribution. The comparison between both method will test the reliability of Prandtl’s Non-linear Lifting Line Theory ability to predict the spanwise lift distribution of HALE UAV ITB.

![Figure 1. First generation of HALE UAV-ITB prototype](image)

2. HALE UAV Configurations

2.1. Basic Configurations
HALE UAV-ITB has high wing and single tail-boom configuration with vertical and horizontal tail as stabilizer. The length is 4 m with 16 m wing span. Airfoil that been used for the wing is EMX 07 and for the stabilizers are NACA 0012.
Table 1. HALE UAV-ITB Configuration

| Specification        | Value  |
|----------------------|--------|
| MTOW                 | 20 kg  |
| Wingspan             | 16 m   |
| MAC                  | 0.4 m  |
| Taper Ratio          | 1      |
| Twist Angle          | 0 degree |
| Aspect Ratio         | 40     |
| S (Wing Area)        | 6.4 m² |
| Altitude             | 20000 ft |

Figure 2. HALE UAV-ITB three view drawing

Figure 3. Airfoil: a.) EMX-07 b.) NACA 0012

2.2. Flight Configurations
For surveillance, the designed HALE UAV ITB will fly at cruise stage as follow:
Table 2. HALE UAV ITB Flight Condition

| Condition         | Value       |
|-------------------|-------------|
| Cruise speed      | 60 km/hour  |
| Angle of Attack (α)| 3 degree    |
| Altitude          | 20000 feet  |
| Air density       | 0.73 kg/m³  |
| Ambient Temperature| 250 K       |
| Ambient Pressure  | 46650 Pa    |

3. Prandtl Non-linear Lifting Line Theory

The following procedure are Prandtl’s Non-linear Lifting Linear Theory method by Anderson[1]. Wing will be divided into a number of k+1 spanwise stations and the initial circulation (Γ) value, at given angle of attack (α), for each station will be assumed as elliptical lift distribution. Then, the induced angle of attack \( \alpha_i \) for each section will be calculated by solving the following equation:

\[
\alpha_i(y_n) = \frac{1}{4\pi V_\infty} \int_{-b/2}^{b/2} \frac{(d\Gamma/dy) dy}{y_n - y}
\]  

(1)

Using effective angle of attack (\( \alpha_{eff} \)) at each station can be calculated by using \( \alpha_i \) from equation (1) and the wing condition angle of attack, by solving the following equation:

\[
\alpha_{eff}(y_n) = \alpha - \alpha_i(y_n)
\]  

(2)

Now, a new \( \Gamma \) distribution is obtained by the latest value of effective angle of attack, and will be compared to the previous value of \( \Gamma \). If a difference occur between the old and new value of \( \Gamma \), a new input value of \( \Gamma \), generated from \( \Gamma_{input} = \Gamma_{old} + D \cdot (\Gamma_{new} - \Gamma_{old}) \) where D is a damping coefficient[1], to perform iteration until the latest value of \( \Gamma \) agree with its previous value within specified margin of error. After that, Kutta – Joukowski theorem, shown by equation (3), can be performed using the \( \Gamma \) value to generate lift at each section.

\[
L'(y_n) = \rho_\infty V_\infty \Gamma(y_n) = \frac{1}{2} \rho_\infty V_\infty^2 c_n(c_l)n
\]  

(3)

Total lift that produced by the wing will be obtained by integrating the value of \( L'(y_n) \) spanwise.

4. Computational Fluid Dynamics (CFD) Simulation

Computational Fluid Dynamics (CFD) will be used as the numerical approach to calculate the spanwise lift distribution of the wing. In this simulation ANSYS CFX will be used as the solver of CFD simulations. CFX solver calculation is based on Reynolds Averaged Navier Stokes (RANS) method, which is derived from Direct Navier Stokes (DNS) method. Instead of Large Eddy Simulation (LES) method, RANS method was choosen because it delivers enough accuracy within faster processing rather than LES, even though it has higher accuracy but takes a lot of memory and running time. The following equations are the governing equations used by ANSYS CFX Solver, including continuity (4), momentum (5) and total energy (7):
Continuity eq.
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \]  
(4)

Momentum eq.
\[ \frac{\partial (\rho \mathbf{V})}{\partial t} + \nabla \cdot (\rho \mathbf{V} \times \mathbf{V}) = -\nabla p + \nabla \cdot \tau + s_M \]  
(5)

Stress tensor, \( \tau \), strain rate relation
\[ \tau = \mu (\nabla \mathbf{V} + (\nabla \mathbf{V})^T) - \frac{2}{3} \delta \nabla \cdot \mathbf{V} \]  
(6)

Total energy eq.
\[ \frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{V} h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{V} \cdot \tau) + \mathbf{V} \cdot s_M + s_g \]  
(7)

Total enthalpy, \( h_{tot} \)
\[ h_{tot} = h + \frac{1}{2} \mathbf{V}^2 \]  
(8)

Shear Stress Transport (SST) method will be used to evaluate the simulation turbulence viscous effect.

4.1. Simulation Model

CFD simulation method is done by the following workflow scheme:

![CFD workflow scheme](image)

**Figure 4.** CFD workflow scheme

The simulation will be simplified as a wing-only half spanwise thus, means actual total lift produced is two times the lift calculated by the simulation. The simulation will be using viscid assumption, to compare the lift distribution result of viscid assumption, by CFD, with inviscid assumption, by Non-linear Lifting Line Theory. Structured mesh blocking method by ICEM CFD will be used to the wing simulation domain. Computational domain that being used are:

1) INLET boundary condition in front of leading edge
2) SYMMETRY boundary condition for wing root side of the domain
3) WALL boundary condition for the wing itself
4) OPENING boundary condition for the other domain.
Figure 5. Side view of generated meshes for wing domain

Figure 6. Computational domains for wing

5. Result

Wing will be divided into 10 section for extracting the CFD results (y = 0, 0.8, 1.6, 2.4, 3.2, 4.0, 4.8, 5.6, 6.4, 7.2, 7.95 m) to obtain Lift/chord (L’) for each station which will be used to obtain the spanwise lift distribution. Chord wise static pressure distribution for each section will be extracted, then by integrating the pressure distribution for upper and lower chord we will obtain L’ for each section.

Here are some notable pressure distribution graph alongside airfoil chord at section y = 0, 7.2, and 7.95 m
Figure 7. Static pressure distribution chordwise at section $y = 0$ m

Figure 8. Static pressure distribution chordwise at section $y = 7.2$ m
By integrating the pressure distribution, lift/chord \( (L') \) for each section will be obtained.

**Table 3. Lift/chord for each section**

| Section | 1  | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 | 11 |
|---------|----|----|----|----|----|----|----|----|----|----|----|
| y (m)   | 0  | 0.8| 1.6| 2.4| 3.2| 4  | 4.8| 5.6| 6.4| 7.2| 7.95|
| L' (N/m)| 18.97| 18.90| 18.88| 18.87| 18.84| 18.78| 18.76| 18.76| 18.60| 17.78| 6.26|

**Figure 9.** Static pressure distribution chordwise at section \( y = 7.95 \) m

**Half Spanwise Lift Distribution**

a.)

![Graph of Static Pressure Chordwise Distribution](image)
By integrating the spanwise lift distribution and times it by two we can obtained the total lift performed by whole wing.

| Total Lift produced and margin of difference by NLLT and CFD method |
|---------------------------------------------------------------|
| Total Lift |
| CFD Method | 288.38 N |
| NLLT Method | 292.59 N |
| Margin of Error | 1.46 % |

6. Analysis and Discussion

There is a slight difference of total lift calculated by CFD and NLLT with very little margin of error, around 1.46%, which is still an acceptable value. The distribution of lift spanwisely, according to Figure 10, shows very little difference distribution besides the wing tip area. Those slight difference are suspected to be caused by mathematically error of the integrating method that been used, which is trapzoidal rule, in order to achive lift/chord for each section. But the trend of distribution is quite the same and still acceptable. It is also due to the simulation that operated in low angle of attack, which is still within the linear region of Cl vs alpha curve thus, lead to more accurate NLLT calculation. Lift distribution differences between both method around wing tip are suspected to be caused by the method of calculating induced drag, which is caused by vortex that created by flow leakage from lower to upper wing. CFD calculation relatively simulate more accurate results towards real condition, due to it’s ability to calculate the static pressure distribution then derive it into induced drag effect to lift distribution. In other hand, NLLT calculate the circulation instead of static pressure distribution which caused less accuracy than CFD calculation. But, the difference between both method is still
small for this case due to the wing configuration that has high aspect ratio and constant chord size without any twisting.

7. Conclusion

The application of NLLT to predict wing spanwise lift distribution for preliminary analysis is recommended for HALE UAV, due to it’s assured accuracy, small margin of error (1.46%) against CFD calculation results, and also requires relatively easier and shorter amount of time to perform.

Acknowledgements

This research was supported by ITB and Direktorat Riset Teknologi dan Pendidikan Tinggi (RISTEKDIKTI).

References

[1] Anderson, John D. *Fundamentals of Aerodynamics (Fourth Edition)*. McGraw-Hill Education, New York. 2009. (Book)
[2] Anderson, John D. *Computational Fluid Dynamics*. McGraw-Hill Education, New York. 1995. (Book)
[3] Carlos E. S. Cesnik, Patrick J. Senatore, Weihua Su, Ella M. Atkins. *X-HALE: A Very Flexible UAV for Nonlinear Aeroelastic Tests*. The University of Michigan. 2010. (Paper)
[4] NACA 0012, http://airfoiltools.com/airfoil/details?airfoil=n0012-il
[5] EMX-07, http://www.aerodesign.de/profile/profile_sp.htm#emx07
[6] Rapinett, Annabel. *Zephyr: A High Altitude Long Endurance Unmanned Air Vehicle*. Department of Physics University of Surrey. 2009 (Paper)