Comparative Study between Schrenk and CFD Analysis for Predicting Lift Distribution along Wing Span of Glider Aircraft

Cahya Amalinadhi Putra, Ratih Julistina, Mochammad Agoes Moelyadi and Taufik Mulyanto

Bandung Institute of Technology, Indonesia
email: moelyadi.mochammad@gmail.com

Abstract: The preliminary analysis of wing of glider aircraftis presented. It gives a decisive consideration for early steps in building a plane. Lift distribution of the wingbecome important consideration to make sure that the generated lift is enough to against its aircraft weight. There are many ways of methods to predict the lift distribution. Theoretical approach is accounted from Schrenk method while more complex calculation is done by Computational Fluid Dynamic method. The objective of this work is to compare the lift distribution between Schrenk and CFD methods. The problem is formulated for high aspect ratio wing that being analyze at 0.0 degree angle of attack and Mach number of 0.07. The result indicates that there is a difference of lift distribution between Schrenk and CFD method.

Key Words: Lift distribution, Static pressure, Schrenk, CFD.

Nomenclature

v : air's velocity
L : lift
L' : lift distribution
p : static pressure
C : chord
c : coefficient
ρ : density
x, y : coordinates
b : span
μ : fluid's viscosity

Subscripts

p : pressure
l : lift

1. Introduction

Predicting the behavior of wing during the early design can efficiently determine the workflow of building a plane. The effect of wing geometric such as washout can easily predicted if only if there is a knowledge of lift distribution. As for the preliminary analysis, the great accuracy of calculation is less needed, on the other hand, the computational time is the most influence part. According to that, researches start the preliminary design of aircraft using quick analysis for predicting lift distribution along span. Among them, shcrenk and CFD analysis become the popular methods.

As a theoretical approach, Schrenk will deliver the curve of lift distribution based on the average between elliptical and trapezoidal distribution[1]. Due to its simplicity of equation, schrenk analysis can give fast overview of lift distribution.

On the other hand, computational fluid dynamics (CFD) analysis requiresgoverning equations such as navier-stokes to be solved to obtain lift distribution along the span. In the CFD simulation, firstly discrete the wing and its computational domain into pieces of small volumes. Having the meshes in the computational domain, solution of flow is resulted by iteratively solved the discrete Navier-Stokes equations. Here, the ANSYS[2] software including ICEMCFD, CFD solver and Post-CFD are used for creating model, meshing, computing and displaying the result.

The aerodynamic forces will be discussed at section 2, while section 3 gives some short and brief explanation about Schrenk and CFD analysis. Next, section 4 describes about how to implement the method to obtain lift distribution. Then, section 5 reports the result of lift distribution. Analysis and comparison between these 2 methods are also provided in this section. Finally, the conclusion of this work is written in section 6.

1. Aerodynamic Forces

The aerodynamic forces are generated when there are flowing fluid exerted on each point of the body in normal and tangential direction[3]. Pressure and shear stress are the basic sources of aerodynamic forces. Both of them can derived from the unit span force components acting on elemental area of the width for the upper and lower surfaces. Moreover, integrating them from the leading edge to trailing edge will obtain the total unit-span forces. The detailed as follow

$$c_p = \frac{p_s - p_\infty}{\frac{1}{2} \cdot p_\infty \cdot v_\infty^2} \quad (1.)$$

$$c_l = \sum \left(c_{p,l} - c_{p,u}\right) \cdot d\left(\frac{x}{C}\right) \quad (2.)$$

$$L' = \frac{1}{2} \cdot \rho_\infty \cdot v_\infty^2 \cdot c_l \cdot C \quad (3.)$$

$$L = \sum L' dy \quad (4.)$$

3. Lift Distribution Approach

There are many methods in obtaining the lift distribution. One of the theoretical approaches is the Schrenk method that generates the average lift distribution as a combination of elliptical and planform lift distribution. On the other hand, CFD method generates pressure distribution on upper and lower surfaces of the wing that can be integrated to obtain lift distribution. The detail of both methods are explained as follow.

3.1. Schrenk Method

One of the most popular analytical method is Schrenk method, proposed by Oster Schrenk[1]. This method become a very good tools for preliminary design of civil aircraft which is given by the Civil Aeronautics Administration. Marcelo[4] has used the following method to obtain lift distribution for early estimation. The idea is done by taking the average lift per unit span between planform lift and elliptical lift. The mathematic models of Schrenk distribution are as follows.

$$L'_{elliptical} = \frac{4L}{\pi b}\sqrt{1-\left(\frac{2y}{b}\right)^2} \tag{5.}$$

$$L'_{planform} = \frac{2L}{(1+\lambda)b}\cdot\left(1+\left(\frac{2y}{b}\right)\cdot(\lambda-1)\right) \tag{6.}$$

$$L'_{Schrenk} = \frac{L'_{elliptical}+L'_{planform}}{2} \tag{7.}$$

3.2. CFD Method

Computational fluid dynamics use numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations that required to simulate the interaction of fluid within the physical bound. The fundamental basis of CFD problems are the Navier-Stokes equations$^{10)}$.

$$\frac{\partial p}{\partial t}+\nabla\bullet(\rho U)=0 \tag{8.}$$

$$\frac{\partial U}{\partial t}+(U\bullet\nabla)U=-\frac{1}{p}\nabla p+F+\frac{\mu}{p}\nabla^2 U \tag{9.}$$

In order to predict the turbulence, the Navier-Stokes equation can be time averaged over a period T time large enough to mean the turbulent fluctuations. The way is by applying the Reynolds decomposition and then time averaging the Navier-Stokes equations. Thus, the Reynolds Averaging Navier-Stokes (RANS) equations are obtained$^{5)}$.

The type of turbulence also become one prominent features in CFD. According to the problem, which is an external flow, the standard $k-\varepsilon$ model is used based on the transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε)$^{6)}$.

4. Method of Implementation

This research is done by following the workflow scheme.

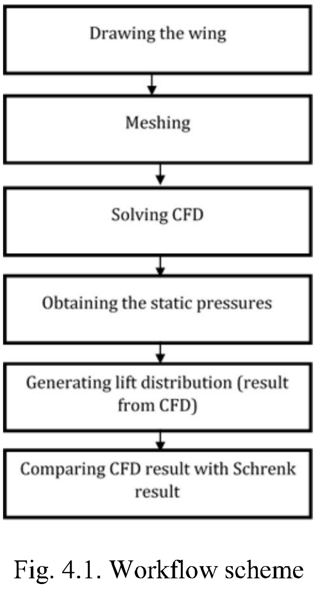

Fig. 4.1. Workflow scheme

The problem case defines a wing with high aspect ratio. A wing configuration of glider is chosen for this research. The configurations are given in Table 4.1.

Table 4.1. Data of wing configuration

Properties	**Value**	**Unit**
Half span	7.1414	[m]
Inner span	4.284	[m]
Chord root	0.933	[m]
Chord tip	0.467	[m]
Area	12	[m^2]
Aspect ratio	17	[-]
Taper ratio inner span	1	[-]
Taper ratio outer span	0.5	[-]

Following the workflow, the first analysis is obtaining the lift distribution through CFD methods. Firstly, the model of wing must be drawn in format of computer-aid drafting (CAD). This process is done with the help of CATIA software$^{7)}$.

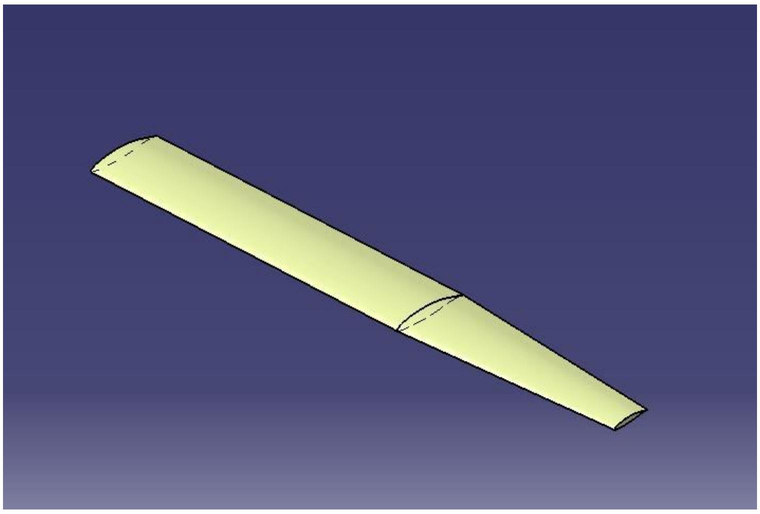

Fig. 4.2. CAD of wing

The CFD solver will need to discrete the model to do the calculation. The discretization uses unstructured grid$^{8)}$. The following grids that used are surface grids and volume grids. In order to catch the flow better around the surface, prism grids are also used$^{9)}$.

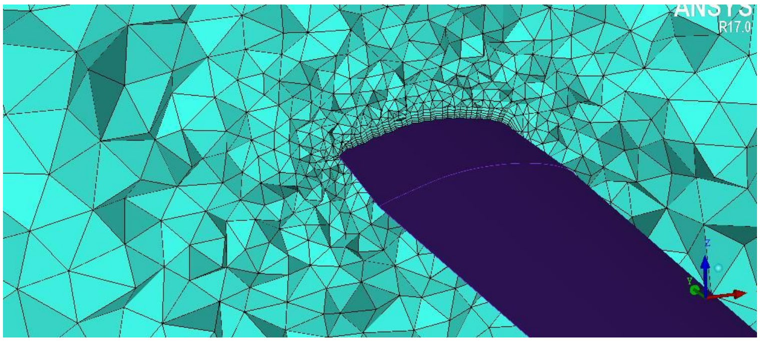

Fig. 4.3. Volume meshing and prism meshing of fluid

The boundary condition for solving are given in Table 4.2.

Table 4.2. Data of boundary condition

Properties	Value
Fluid type	Air, ideal gas
Mach number	0.07 [-]
Turbulence type	$k-\varepsilon$
Error	10^{-4}

The result that obtain from CFD solver is not a distribution of lift, but the distribution of pressure. However, the lift distribution still can be determined by the following steps that using the given static pressure.

1) Divide the span into several wing sections. The sections are placed at 20%, 40%, 60%, 80% and 95% of span, respectively.
2) Get the value of lift per unit of span (L') for each wing section.
3) Calculate the resultant of static pressure distribution for each wing section along span.
4) Find the correlation of the production of lift of several section from step 1) and the difference value of static pressure obtained from step 3)
5) Produce lift distribution along span by doing regression with the result from step 4)

5. Result and Analysis

The aerodynamic data that produced by CFD at 0.0 degree angle of attack and Mach number of 0.07 is shown in the table 5.1.

Table 5.1. Aerodynamic result of wing.

Properties	Value	Unit
Lift	3,258.34	[N]
c_l	0.7498	[-]

Following the steps of obtaining lift distribution and implementing Eq. (1.) and Eq. (2.), these are the static pressure distributions over chord in several wing section along the span, the static pressure distribution along the span, and data of lift coefficient per unit span.

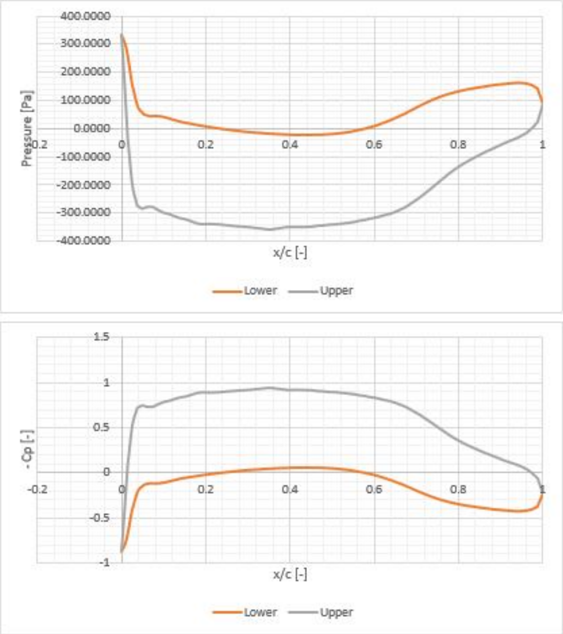

Fig 5.1. Pressure and Cp distribution along chord in 20% span

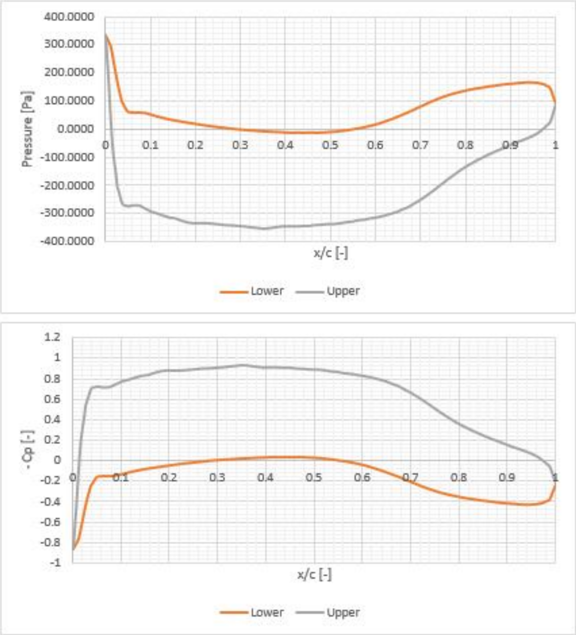

Fig 5.2. Pressure and Cp distribution along chord in 40% span

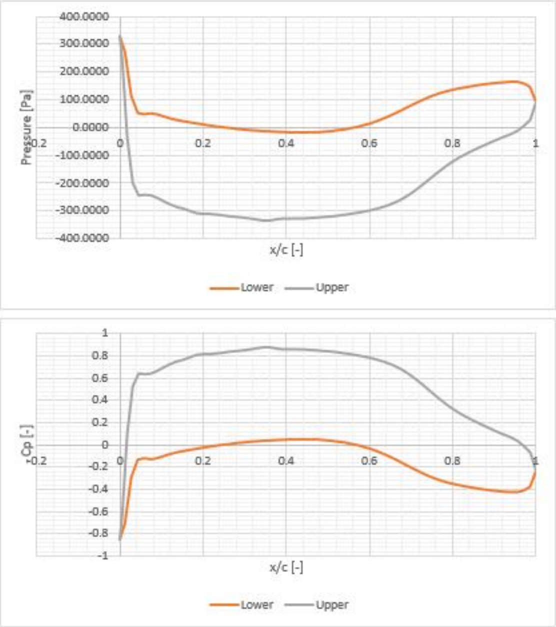

Fig 5.3. Pressure and Cp distribution along chord in 60% span

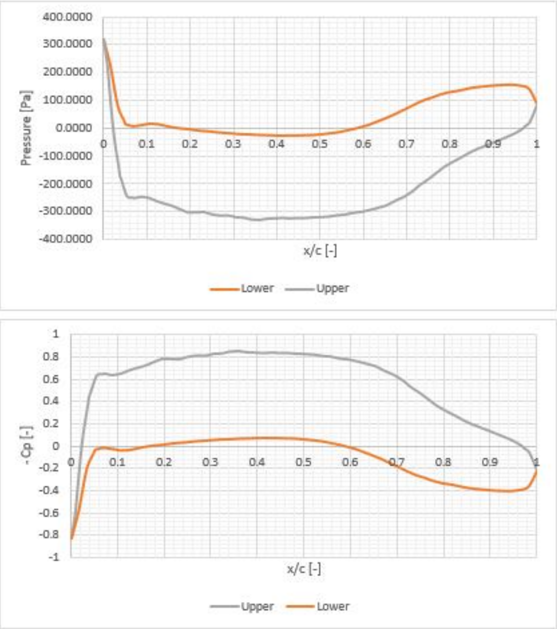

Fig. 5.4. Pressure and Cp distribution along chord in 80% span

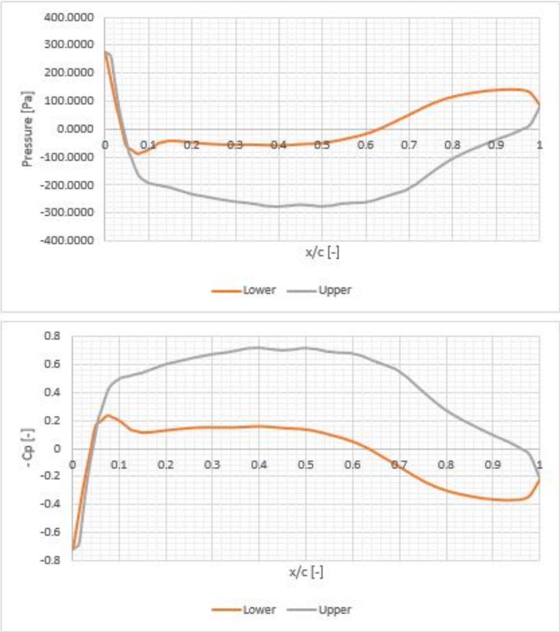

Fig. 5.5. Pressure and Cp distribution along chord in 95% span

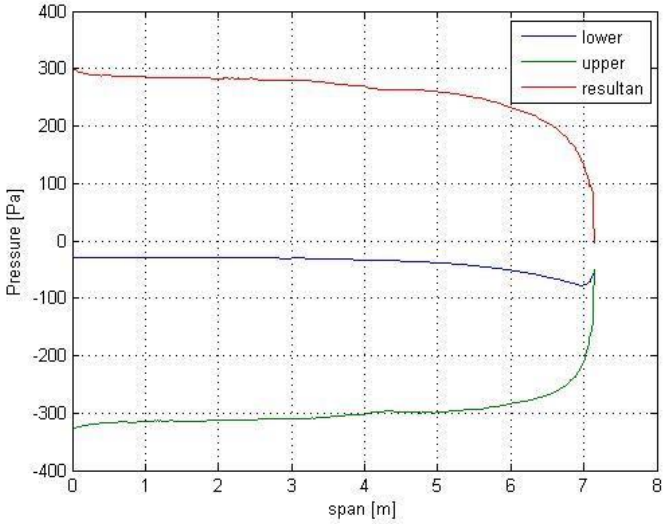

Fig. 5.6. Pressure distribution along span

Table 5.2. Data of lift coefficient and lift per unit span

Location [% span]	**Cl [-]**	**L' [N/m]**
20	0.7455	285.394
40	0.7421	284.081
60	0.6901	264.178
80	0.4784	183.121
95	0.2509	96.028

Fig. 5.1. to 5.5. shows the characteristics of static pressure over the chord. The upper part is likely more negative than the lower part. From Fig. 5.4. to 5.5., the absolute static pressure of the upper part becomes smaller due to the change of chord's size.

Based on step 3) to 5), it is very important to create the relation between lift distribution, static pressure, and chord to obtain the derived distribution. Following Eq. (1.) to Eq. (3.), the relation can be written as a simple mathematical equation as follows.

$$L' = k \cdot \Delta p_s \cdot c \tag{10.}$$

Following Eq. (10.), obtained the value of k.

Table 5.3. Comparison of k

Location [% span]	k **[-]**
20	0.9103
40	0.9156
60	0.8968
80	0.8867
95	0.8493

From Table 5.3., the average value of k will be, $k = 0.8918$. Implementing value of k to Eq. (10.), the error between derived lift per unit span from Eq. 10 and CFD lift per unit span given as follows

Table 5.4. Comparison of k

Location [% span]	**L' CFD [N/m]**	**L' derived [N/m]**	**Error [%]**
20	285.394	279.571	2.04
40	284.081	276.698	2.59
60	264.178	262.693	0.56
80	183.121	184.159	0.57
95	96.028	100.82	4.99

Following the Eq. (4.), obtained the value of error between derived lift with CFD about 0.09% which is still small. Therefore, the derived lift per unit span based on k and Eq. (10.) can be interpolated over the span and produce the lift distribution over the span.

On the other hand, by following Eq. (5.) to Eq. (7.), the lift distribution from Schrenk method can be obtained. The value of lift that used is taken from the CFD result as shown in Table 5.5. ant the comparison of lift distribution along span between Schrenk and CFD methods is depicted in Fig. 5.7.

Table 5.5. Aerodynamic result of wing

Properties	**Value**	**Unit**
Lift CFD	3,258.340	[N]
Lift Derived	3,261.284	[N]
Lift Schrenk	3,258.340	[N]

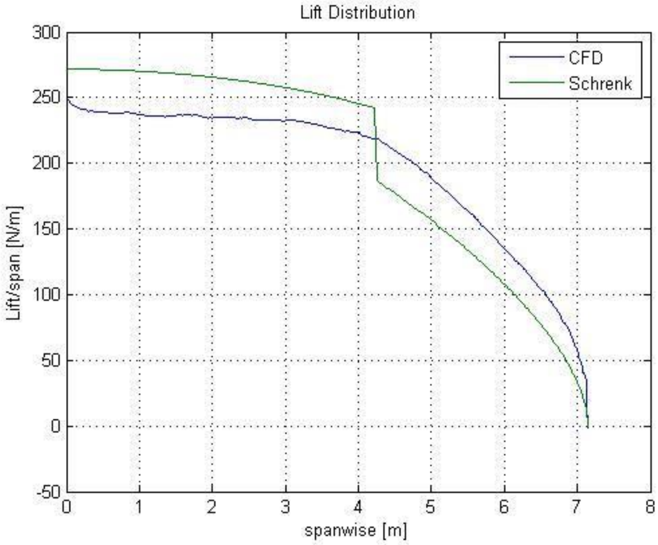

Fig. 5.7. Comparison of lift distribution between CFD and Schrenk method

According to Fig. 5.7, there is a difference between Schrenk and CFD results. Within the inboardwing, Schrenk analysis have a higher value of lift per unit span than CFD method. On the other hand, the value of lift per unit span of schrenk is lower than CFD in the out boardwing.

That differences is due to the limitation of Schrenk method. From section 3, the form of graph from CFD is derived from pressure distribution all over the wing, while the graph from Schrenk method is derived according to planform of wing, which is affected by size of chord. Moreover, the sharp difference between inboard and outboard wing is due to the change of value of taper ratio.Those results indicate the error of CFD result is 15% from Schrenk results.

6. Conclusion

There is a difference between CFD method and Schrenk method in obtaining lift distribution. The difference comes from the basic methods that are being used. CFD used the integration of static pressure over the wing, while Schrenk imitate the average of elliptic and planform distribution over the wing.Those results indicate the error of CFD result is 15% from Schrenk results.

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

References

1) Schrenk, O.: *A Simple Approximation Method for Obtaining the Spanwise Lift Distribution, NACA TM 948*, 1940.
2) Nguyen Minh Triet, Nguyen Ngoc Viet, Pham Manh Thang.*: Aerodynamic Analysis of Aircraft Wing, VNU Journal of Science: Mathemathics,*Hanoi, Vietnam, pp. 68-75,2015.
3) Abbot, I. H. and von Doenhoff, A. E.: *Theory of Wing Sections*, 1959
4) Tamura, Satoru, Marcelo, Flávio Luiz de Silva Bussamra: *A Simplified Geometric Method for Wing Loads Estimation*, Brazilian Symposium on Aerospace Eng. & Applications, 2009.
5) Stoll, Alex M.: *Comparison of CFD and Experimental Results of the LEAPTech Distributed Electric Propulsion Blown Wing*, Joby Aviation, Santa Cruz, California. 2015.
6) Crasto, Giorgio: *Numerical Simluations of the Atmospheric Boundary Layer*, Università degli Studi di Cagliari, 2007.
7) Pepelea, Dumitru, Cojocaru, Marius G., Toader, Adrian, Niculescu, Mihai L.: *CFD Analysis for UAV of Flying Wing*, National Aerospace Research, Bucuresti, Romania, 2016
8) Demir, Gorkem, Alemdaroglu, Nafiz: *CFD Solutions to Store Separation Problem*, Ankara International Aerospace Conference, 2015.
9) Justin Petrilli, Ryan Paul, Ashok Gopalarathnam,: *A CFD Database for Airfoils and Wings at Post-Stall Angles of Attack*, American Institute of Aeronautics and Astronautics, 2014
10) J.D., Anderson, G., Degrez, E., Dick, R., Grundmann: *Computational Fluid Dynamics*, 1959