Optimization of the Aerodynamic Efficiency of Micro Aerial Vehicle Wings using Rapid Prototyping Techniques

David E. Ghati

University of New South Wales at the Australian Defence Force Academy

Research into the low Reynolds (Re) number flight of Micro Aerial Vehicles (MAVs) has increased due to the expectancy of deployable assets for use in confined urban environments. This is for both military and civilian applications for intelligence, surveillance and reconnaissance (ISR) purposes. This report details the research conducted to identify and implement rapid prototyping techniques to optimize the aerodynamic efficiency of the MAV wing. The wing geometry is optimized by maximizing the lift to drag ratio (L/D) of an aerofoil shape, taking particular interest in the variation of the wing sweep (\(A\)) and twist (\(\phi\)). The wings analysed are confined to the sizing requirements for MAV classification restrictions of a maximum dimension of 150 mm and are expected to fly at speeds of less than 15m/s. This report presents the results of the MAV wing testing and analysis using mathematical computational fluid dynamics for the rapid prototyping analysis required. Testing was completed in FLUENT, with an ideal mesh developed for later use in the wing development. Wing sweep was tested for angles between 30° forward and 55° back at increments of 5°, while twist was tested at increments of 1° for all angles between -5°\(\phi\) and 15°\(\phi\). Optimization results were collaborated with a second study which studied the effects of taper ratio (\(\lambda\)) and thickness to chord ratio (\(t/c\)). An optimized wing is proposed with \(\phi = 9°\) twist \(A = 0°\) on a NACA0008 airfoil (\(t/c = 8%\)) with \(\lambda = 0.5\).

Contents

I. Background .................................................................................................................................................. 3
   A. Introduction ........................................................................................................................................... 3
   B. Flight at Low Reynolds Numbers ........................................................................................................... 4
   C. Current MAV Designs ............................................................................................................................ 5

II. Effect of Geometric Aerofoil Parameters on Performance ........................................................................... 6
   A. Trailing Vortex Induced Drag and the influence of the Aspect Ratio ...................................................... 6
   B. Sweep .................................................................................................................................................... 7
   C. Wing Twist ............................................................................................................................................ 8
   D. Aerofoil Thickness to Chord Ratio ......................................................................................................... 8
   E. Wing Taper Ratio .................................................................................................................................. 9
   F. Other Considerations .............................................................................................................................. 9

III. Rapid Prototyping Using Computational Fluid Dynamics ......................................................................... 10
   A. Result Accuracy, Precision and Reliability ............................................................................................ 10
   B. Computational Fluid Dynamics ............................................................................................................. 10

IV. Project Overview ...................................................................................................................................... 11
   A. Project Intent ......................................................................................................................................... 11
   B. Methodology ....................................................................................................................................... 11

V. Initial CFD Setup ...................................................................................................................................... 12

1 OFFCDT, School of Engineering & Information Technology, Aeronautical Engineering Thesis and Practical Experience ZEIT 4500/4501 & CDFSP Elective 2 ZEIT4010
A. Initial Setup Overview........................................................................................................... 12
B. Wing Generation ..................................................................................................................... 12
C. Computational Fluid Dynamics Analysis using ANSYS 12 Workbench.............................. 13
D. Mesh Refinement Study .......................................................................................................... 15
E. Characteristics of Mesh Reliability Results........................................................................... 17
F. Verifying Results using Prandtl's Lifting Line Theory............................................................ 18
VI. Initial Testing Results........................................................................................................... 18
A. Initial Testing Overview ......................................................................................................... 18
B. Twist Testing ......................................................................................................................... 19
C. Sweep Testing ....................................................................................................................... 22
D. Mesh re-evaluation .............................................................................................................. 24
E. First Refined Mesh (Mesh 2).................................................................................................. 25
VII. New Mesh............................................................................................................................ 27
A. New Mesh Design .................................................................................................................. 27
B. Validating Mesh Changes ....................................................................................................... 28
C. New Testing ........................................................................................................................ 30
D. Finest Mesh .......................................................................................................................... 32
VIII. Collaboration ..................................................................................................................... 33
IX. Wing Manufacturing and Wind Tunnel Testing ................................................................. 34
X. Conclusions .......................................................................................................................... 35
XI. Recommendations .............................................................................................................. 35
Acknowledgements .................................................................................................................. 36
References .................................................................................................................................... 37

APPENDICES (In supplementary document)

Nomenclature

\( \alpha \) = angle of attack [deg]  
\( \alpha_0 \) = calculated 2D lift curve slope [deg\(^{-1}\)]  
\( \alpha(\theta_0)/\alpha_0(\theta_0) \) = calculated 3D/2D lift curve slope at angular displacement \( \theta_0 \) [deg\(^{-1}\)]  
\( \beta \) = viscosity ratio  
\( \epsilon \) = uncertainty [%]  
\( \lambda \) = wing taper ratio  
\( \Delta \phi/\Delta \phi_{rad} \) = wing sweep angle in degrees/degree wing sweep in radians [deg/deg/rad]  
\( \rho \) = fluid density [kg/m\(^3\)]  
\( \tau_w \) = shear stress at the wall [Pa]  
\( \alpha(\theta_0)/\phi(\theta_0) \) = wing twist/ degree wing twist/ wing twist at angular displacement \( \theta_0 \) [deg]  
\( AR \) = wing aspect ratio  
\( b \) = wingspan [mm]  
\( c/c(\theta_0) \) = chord / chord at angular displacement \( \theta_0 \) [mm]  
\( C_l/C_{l_{max}} \) = coefficient of lift / maximum coefficient of lift  
CFD = Computational Fluid Dynamics  
\( D/D_i \) = drag/profile drag/induced drag [N]  
\( I \) = turbulence intensity  
\( k \) = empirical correction factor based on the aerofoil used in the analysis (Raymer, 2006)  
\( (L/D) \) = aerodynamic efficiency, lift on drag ratio  
\( N/N_c \) = mesh cell density or number/ Xth Mesh
\[ P \] = order of convergence \\
PLL\( T \) = Prandtl’s Lifting Line Theory \\
\( Re \) = Reynolds number \\
\[ u/y-u'/\bar{u} \] = velocity (vel.) / wall friction vel. / RMS turbulent vel. fluctuations / \( Re \) average mean vel. [m/s] \\
\[ \nu / \nu_i \] = local fluid kinematic viscosity (laminar/turbulent) [m²/s] \\
y = distance to nearest wall [mm] \\
y⁺ = non-dimensional wall distance

I. Background

A. Introduction

The past decade has seen a sudden increase in interest and investment by the aviation industry into miniaturized flight due to its many possible civil and military applications (see Appendix A). This new class of autonomous, lightweight, small-scale flying machines are called Micro Aerial Vehicles (MAVs). Imagined to be a ‘small speck in the sky that approaches in virtual silence, unnoticed’ (McMichael, et al., 1997), there is a certain mythical aura placed around these MAVs by many perspective operators. MAVs are expected to be introduced into a myriad of different service operations, feeding off the current success of current larger drones and unmanned aerial vehicles (UAVs) that have redefined military ISR in recent history. Due to their relatively low cost and their tendency to provide accurate intelligence, numerous UAV development programs have been initiated worldwide (Pines, et al., 2006; Raney, et al., 2004). Hence, by concentrating on the small scale, MAVs are expected to be virtual scouts, disposable team members that are used to explore the dangerous urban environment to provide indispensable information without placing human team members in unnecessary danger (Tamai, et al., 2008; Pines, et al., 2006). By using these small fliers and attaching simple off-the-shelf sensors and cameras, MAVs will be used to check for hazards such as chemical weapons, booby traps, fires or even enemy forces. Other proposed applications also include scouting for injured or surviving personnel in unstable buildings, or even sensing ‘the trace of a suspected chemical agent and deploy[ing] a small tagging device, attaching it to the vehicle’ before ‘vanishing down a narrow alley’ (McMichael, et al., 1997).

Due to recent technological advances and research, the concept of small, rapidly deployable ‘eye in the sky’ autonomous flight vehicles has turned from fiction into a reality (Iifu, et al., 2001). Continual reductions in size, weight and power consumption are improving the feasibility of MAVs for practical use as inexpensive and expendable platforms where larger vehicles are not practical (Ifju, et al., 2001). Alongside this increased research into miniaturized technologies, this current study is designed to analyse the effects that aerofoil geometry has on the aerodynamic efficiency of the wing. Hence, by optimizing it, this will ensure an efficient design that will require less power to operate, thereby further reducing the aircraft size and weight. This investigation is one of several current studies currently being undertaken to develop a school MAV.

There are two predominate classes of MAVs, the 24” and 6”, with several variants of the definitions of the particular constraints to each class, and with the latter class being of particular interest to this study. Generally, (see Table 1) this class of MAVs is characterized by a maximum length of 6 inches (150 mm), travels at flight speeds of less than 15 m/s, and has a maximum gross takeoff weight of approximately 200 g (Tamai, et al., 2008; Pines, et al., 2006; Raney, et al., 2004; Shyy, et al., 1999; Stanford, et al., 2007). In the United States, the development of these vehicles has been led by the Department of Defense’s (DoD) need to develop small flying machines that are able to accomplish reconnaissance missions in confined spaces (Pines, et al., 2006). The Defense Advanced Research Projects Agency (DARPA) developed a set of performance criteria for a MAV workshop which required designs to put on it another constraint of expected mission duration at double the current 20 to 30 min while carrying a payload of 20 grams or less to a distance of perhaps 10 km (Shyy, et al., 1999). Furthermore, the competition required the MAV to observe a target located 600 m from the launch site and to keep a two-ounce payload aloft for 2 minutes (Grasmeyer, et al., 2001). In order to accommodate all these requirements for this emerging sector in the aerospace market, advancements in miniaturized digital electronics, low Reynolds number (Re) aerodynamics, multidisciplinary design methods and other enabling systems technologies for MAVs became required (Tamai, et al., 2008).

This research took particular interest in the effect of four geometrical parameters on the aerodynamic efficiency of a given MAV aerofoil, namely the wing taper ratio (\( \lambda \)), thickness ratio (\( t/c \)), wing twist (\( \phi \)) and...

<table>
<thead>
<tr>
<th>Specification</th>
<th>MAV Design Requirements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Size</td>
<td>150 mm (6in)</td>
</tr>
<tr>
<td>Weight</td>
<td>100 - 200 g</td>
</tr>
<tr>
<td>Range</td>
<td>1 to 10 km</td>
</tr>
<tr>
<td>Endurance</td>
<td>60 min</td>
</tr>
<tr>
<td>Altitude</td>
<td>150 m</td>
</tr>
<tr>
<td>Speed</td>
<td>15 m/s</td>
</tr>
<tr>
<td>Payload</td>
<td>10 - 20 g</td>
</tr>
<tr>
<td>Cost</td>
<td>$1,500</td>
</tr>
</tbody>
</table>

Table 1. MAV Design specifications (Pines, et al., 2006; Shyy, et al., 1999)
wing sweep ($\delta$). The latter two parameters will be analysed in greater depth in this analysis, whereas the former two parameters have been studied in a similar concurrent research project undertaken by James Walduck alongside this project. The methodology undertaken in this study is the use of rapid prototyping techniques through the use of Computational Fluid Dynamics (CFD) analysis and low speed wind tunnel testing.

B. Flight at Low Reynolds Numbers

There have been several MAVs developed recently, using both fixed and biologically inspired wing designs (Raney, et al., 2004; Stanford, et al., 2007). The predominate method employed in MAV design is the use of scaling factors to reproduce the characteristics of the conventional aircraft onto the miniature scale primarily based on the $Re$. This enables the characterization of the aerodynamic performance of the wing (Ifju, et al., 2006). Theoretically, two geometrically similar aerofoils with the same relevant dimensionless value have dynamically similar flow geometries, even if they are flowing in different fluids with different flow rates (Anderson Jr, 2008). However, although $Re$ is linearly proportional to the characteristic length, conventional flight occurs with the $Re$ magnitude in the region of $10^5$ – $10^6$ whilst MAVs, due to their smaller size, will have $Re$ in the range of $10^2$ to $10^3$. Due to this difference of several orders of magnitude the traditional aerodynamic design principles used for conventional aircraft cannot be used for MAVs. In this $Re$ region, the aerodynamic performance, particularly for smooth wings, are dramatically reduced as seen in Figure 1 (Ifju, et al., 2001), due to the presence of the separation bubble in this regime (Vieriu, et al., 2006). The laminar flow that dominates the flow around the aerofoil is easily separated even across the short distance along the chord-line of an MAV wing, causing bubbles to form before transitioning and reattaching as turbulent flow. Consequently, this dramatically affects the performance of the lifting surface (Pines, et al., 2006; Mueller, et al., 2003). As can be seen from Figure 2, as the angle of attack ($\alpha$) increases, this bubble continues to travel up towards the leading edge of the wing before bursting, (Figure 3) causing a further reduction in lift (Simons, 1994).

Noting that the low $Re$ regime is typical to the realm of large insects and small birds (Shyy, et al., 1999), failure to understand the effects of flying in this regime effectively makes mechanised flight impossible (Dickinson, et al., 1999). In nature, the relationship between $Re$ and aerodynamic efficiency is observed, where large species ($Re>100,000$) soar for extended periods of time at high wing loadings. This is comparable to smaller birds such as the hummingbird, which is more akin to the size of a MAV. The hummingbird flaps vigorously at high frequency to remain airborne (Ifju, et al., 2001), expending far more specific energy in comparison. Thus, one conclusion is that the MAV size parameters require high surface-to-volume ratios to ensure flight within severely constrained weight and volume limitations (Ifju, et al., 2001). However, this takes away from the ability to use efficient high aspect ratio (AR) wings. Rather, more panned out - almost circular - designs are preferable, as they maximize the lifting area.

Another major setback in MAV flight is the reduced stability and control of the aircraft due to the small mass moments of inertia and also the local turbulence naturally exhibited in the atmosphere at such low altitudes which is comparable to the flight speed of these vehicles. Consequently, the airspeed variations over the MAV wing will be proportionally larger relative to the flight speed and will be unbalanced across the wingspan leading to control difficulties in either remote operation or using onboard autopilots (Ifju, et al., 2001). This is due to issues with the short response times required to ensure stability. Small variations in airspeed across the wing will inevitably cause the wing to experience large differences in $Re$ and, particularly in the range of $10^5$ as

![Figure 1. Change in observable maximum $L/D$ with respect for $Re$ (Ifju, et al., 2001; Lissaman, 1983)]

![Figure 2. Typical flow around an MAV with the laminar bubble forming before turbulent flow reattaches for (a) low $\alpha$ and (b) high $\alpha$. (Simons, 1994 pp. 37, Fig. 3.8 & 3.9)]
shown in Figure 1, a great change in the experienced aerodynamic efficiency (Santhanakrishnan, et al., 2005). Furthermore, thin aerofoils at low Re are sensitive to pitch due to their steep lift curve slopes. Hence, small gusts that alter the local $\alpha$ may cause a sudden wing pitch up, causing stall, or pitch down, which in turn causes reduced lift (Simons, 1994). Both of these scenarios cause dangerous consequences to the aircraft and must be countered. One possible solution is the use of rough aerofoil surfaces as they experimentally have shown to perform better at low Re flight (Santhanakrishnan, et al., 2005) since they appear to negate the sudden transition in $L/D$. However, this does provide a lower overall maximum $L/D$ in the upper region of MAV flight where the aircraft is expected to predominately perform during its mission.

C. Current MAV Designs

Most current fixed wing MAVs are of generally a similar design, predominately due to the dimensional limit on the aircraft forcing the use of inefficient low AR designs. Here, a quick overview of two particular designs is given, both of which were initially designed to the standards of the DARPA competition and have been extensively developed over the past decade. A further, more succinct and detailed summary of a wider range of both fixed wing and rotor MAVs is provided in Appendix B, noting the high $L/D$ achieved by these aircraft ($L/D_{\text{max}} \approx 5-6$).

1. University of Florida MAV

Nicknamed the ‘Gators’, The University of Florida (UF) have designed a range of MAVs that have a distinct design distinguishable by the bat-like wing adopted to take advantage of flexible membrane wing design with an under-cambered aerofoil (Figure 4-5). The wing consists of a latex rubber membrane skin structure with a carbon fibre skeleton support, in order for the wing to not only be of the most lightweight design but also to have a passive shape adaption feature that allows the wing to deform slightly with the oncoming flow and hence reduce drag (Ifju, et al., 2001). The design has been in continuous development since 1997, continually adapting its flexible wing design and undergoing research into the effects and performance through extensive testing and analysis (Ifju, et al., 2006). The UF MAV has shown great promise, winning numerous awards for its performance in national competitions (Abdulrahim, et al., 2002).

2. AeroVironment - Black Widow/Wasp

The Black Widow (Figure 6(a)) was the first attempt by AeroVironment to produce a prototype for the DARPA competition, to explore the 150 mm aircraft design space (Parsch, 1997). This first generation design used a solid foam wing structure with internal reinforcements with the avionics and payload embedded into the wing. The design concentrated on the autonomous nature of the MAV, building this feature into the aircraft with a colour camera with downlink transmitter, and a payload weight of 80 grams. It achieved ranges of 1.8 km and endurance of 30 mins (Grasmeyer, et al., 2001). However, it appears that this project has ended, with
AeroVironment being contracted to continue development of MAVs for the U.S. military, developing the Black Widow’s successor, the Wasp MAV, since 1998 (Parsch, 1997).

The Wasp is a hand-launched, recoverable flying-wing UAV, which uses synthetic materials that act both as a battery and main wing structure (Figure 6(b)). It is waterproof and is recovered automatically by a horizontal landing on land or water. The UAV can fly for up to an hour, is equipped with a GPS-based navigation system for fully autonomous missions and has a payload of several miniature video cameras. The original developmental Wasp has continually been enhanced with the development of the larger Block II which set the MAV record of 107 min endurance and the Block III (Figure 6(c)) which is the current U.S. Air Force BATMAV (Battlefield Air Targeting Micro Air Vehicle). The Wasp Block III is notable due to the significantly different design from previous variants with extra wing panels and modified fuselage. It is incomparable to its Black Widow origins in shape or size (Parsch, 1997), failing to conform to any of the dimensional requirements of MAV design (see Table 2).

![Figure 6. Chronological development of the AeroVironment Black Widow/Wasp UAV design. (a) Black Widow, (b) Wasp Block I and (c) Wasp Block III (Grasmeyer, et al., 2001 pp. 1, Fig. 1; Parsch, 1997)](image)

<table>
<thead>
<tr>
<th>Black Widow</th>
<th>Wasp Block I</th>
<th>Wasp Block II</th>
<th>Wasp Block III</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
</tr>
<tr>
<td></td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
</tr>
<tr>
<td>Wingspan</td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
</tr>
<tr>
<td></td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
<td>15 cm (6 in)</td>
</tr>
<tr>
<td>Weight</td>
<td>56.5 g (0.12 lb)</td>
<td>170 g (0.37 lb)</td>
<td>275 g (0.61 lb)</td>
</tr>
<tr>
<td></td>
<td>41 cm (1.1 miles)</td>
<td>40-60 km/h (25-37 mph)</td>
<td>40-65 km/h (25-40 mph)</td>
</tr>
<tr>
<td></td>
<td>33 cm (13 in)</td>
<td>40-60 km/h (25-37 mph)</td>
<td>40-65 km/h (25-40 mph)</td>
</tr>
<tr>
<td>Speed</td>
<td>1.8 km (1.1 miles)</td>
<td>4 km (2.5 miles)</td>
<td>5 km (3.1 miles)</td>
</tr>
<tr>
<td></td>
<td>33.4 min.</td>
<td>40-60 min.</td>
<td>45 min.</td>
</tr>
<tr>
<td>Propulsion</td>
<td>Electric motor; 4.65 W</td>
<td>Electric motor; 10 W</td>
<td>Electric motor; 10 W</td>
</tr>
</tbody>
</table>

Table 2. Technical data of the Black Widow/Wasp design. (Grasmeyer, et al., 2001; Parsch, 1997)

II. Effect of Geometric Aerofoil Parameters on Performance

A. Trailing Vortex Induced Drag and the influence of the Aspect Ratio

Due to the difference in pressure between the lower and upper surfaces of the wing, lift is generated (Anderson Jr, 2008). However, it is this phenomenon that causes wing tip vortices, because at the tips the higher pressure air flows outward curling around the wing onto the upper surface. This causes an upwash on the wing edges which forms a vortex behind the wingtip that trails off. This effectively generates a downwash effect behind the wing as the vortex curls over inboard. Although this does not produce any useful lift, it defines the airflow direction above the wing as seen in Figure 7 (Simons, 1994).

![Figure 7. Schematic of the downwash effect.](image)
This phenomenon causes a reduction in the effective aerodynamic $a$ of the aircraft as it causes a rotation of the aerofoil relative to the oncoming flow. Hence, the wing reaction force is no longer solely lift generation but a component of this force generated by the aerofoil acts in the aft direction as a drag. Therefore, if the wing had no tips (two-dimensional aerofoil) this effect wouldn't occur; however, by increasing the wing $AR$, this effect can be reduced (Raymer, 2006).

For highly efficient aircraft, a high $AR$ is ideal in that it dramatically reduces the vortex induced drag, particularly at low speeds at higher lift coefficients ($C_L$) (Simons, 1994). This is evident in nature (Ifju, et al., 2006), as birds such as the albatross with high AR wings ($AR = 15$) fly at low speeds and high lift coefficients ($C_L=1$). Conversely, although more manoeuvrable, the lower AR Petrel wing ($AR = 8$) forces it to proceed primarily by flapping or flap-gliding due to its comparatively lower $C_L$ of only 0.25 (Chklovski, 2010). Hence, high AR coupled with high wing loading will ensure good soaring performance and high penetration at reduced flight speed (Simons, 1994).

Increasing the AR generally causes the wing to reach not only a higher $C_{L\text{ max}}$ but the lift curve slope is increased and hence the $C_{L\text{ max}}$ is achieved at lower angles of attack ($\alpha$). However, as evident in Figure 8, the $C_{L\text{ max}}$ is a function of the boundary layer flow and hence an increase between moderate to high AR will not increase the $C_L$ achieved (Simons, 1994). The direct involvement of the AR on the vortex induced drag ($C_{D_{i}}$) of the aerofoil is evident in the standard equation:

$$C_{D_{i}} = k \frac{C_{L}^{2}}{\pi AR}$$

(1)

However, it must be noted that there are not limitations on the amount of gain due to increased $AR$. Through the $AR$, particularly for a thin aerofoil at low $Re$, this will increase the pitch sensitivity of the wing, decreasing the overall predictability and controllability of the aircraft with even small gusts.

Another issue is that to increase the $AR$, the chord will become too small, and the $Re$ of the wing will decrease to the extent that efficient flight would become impossible (Raymer, 2006). Furthermore, for design requirements for the MAV, there is a dimensional limit which must be adhered to. Hence, the increased $AR$ will incur a reduction in surface area below what is required to achieve flight. For this reason, as can be seen from previous designs, most current MAVs employ very low $AR$ in order to achieve the required lift. Other advantages of utilizing a low $AR$ wing profile include the ability of the aircraft to fly safely at a wider range of angles, and that it is easier to trim and less sensitive to gusts. However, this does incur the penalty of a lower $C_{L \text{ max}}$ as the wing becomes virtually engulfed by the effects of the wingtip vortices and the strong cross flows reduce the effectiveness of the wing. It becomes evident however, when the dimensional restriction is increased, as with for the larger 61 cm (24") MAV convention, the use of higher $AR$ wing profiles become dominant, as seen in Figure 9, which depicts comparisons of the two UF Gator MAVs and the continuous development of the AeroVironment Black Widow/Wasp design. It is therefore evident that the effects of $AR$ on small scale wing design have been exhaustively tested. Consequently, such effects were not be investigated in this report, but rather an arbitrary $AR$ of 2.0 was chosen for the control wing and is used for the rest of the research.

B. Sweep

Wing sweep ($\Lambda$) is generally used on full sized aeroplanes because they are beneficial for flight approaching the speed of sound. Hence, pronounced $\Lambda$ in small aircraft is usually employed only to model a larger fighter
aircraft. However, it must be noted that it is more important to this investigation that although \( A \) is rarely used on low speed aircraft, tailless aircraft such as most current MAV designs, commonly employ backward \( A \) for stability. However, to counter the negative effects of control at low speeds, employing back \( A \) may require the use of special devices such as slots or boundary layer fences (Simons, 1994).

There are several slight advantages in employing \( A \). Forward \( A \) particularly aids control at low speeds and delays wing tip stall. However, although back \( A \) has the opposite effect on control at low speeds, it causes a slight dihedral effect and may be justified for aerobatic designs as it provides steadier flight both inverted and right side up (Simons, 1994).

Therefore, considering that small amounts of \( A \) have little effect on vortex drag (Simons, 1994), it is of interest for this research to test the aerodynamic efficiency of \( A \) to determine whether the advantages stipulated can be used in the design. For this research \( A \) was tested at increments of five degrees up to the maximum dimensional limit of 55°/A aft and 30°/A to ensure any advantage in \( L/D \) is captured.

C. Wing Twist

![Figure 10. Variation in camber and twist in a pigeon’s wing (Chklovski, 2010).](image)

Wing twist (\( \varphi \)) is used to prevent tip stall and adjust the lift distribution to approximate that of an elliptical aerofoil, by altering the local \( \alpha \). There are three general forms of \( \varphi \) that can be applied systematically to the aerofoil - geometric, linear and aerodynamic twist. A wing will have washout when the tip aerofoil is at a nose-down angle relative to the wing root which tends to promote stall there first, improving control during stall. Similarly, wash-in is defined as when the tip has a nose-up angle and has the opposite effect on the stall characteristics of the aerofoil.

Reshaping the lift distribution by manipulating \( \varphi \), the change in lift at a point along the wing is proportional to the ratio of the new \( \alpha \) to the original one. Hence, as the change in lift distribution is dependent on the original \( \alpha \), the \( \varphi \) optimization is only valid at one lift coefficient and, consequently, at a different \( \alpha \), the given \( \varphi \) will not provide the whole optimization benefit. Therefore, generally, the more \( \varphi \) required for optimization at a particular lift coefficient, the worse the wing performance at other coefficients; and hence why \( \varphi \) is generally kept to a minimum.

Furthermore, \( \varphi \) will also change the spanwise lift distribution due to the change in local \( \alpha \). For this reason, \( \varphi \) is rarely applied more than \( \pm 3^\circ \) (Simons, 1994). However, it is evident that in nature, higher \( \varphi \) are employed, as seen in the pigeon wing which has \( \varphi = 13^\circ \) as shown in Figure 10 (Chklovski, 2010).

In all current MAV designs, \( \varphi \) does not appear to be utilized and is not discussed in the design analysis of the MAVs studied above. For this research, \( \varphi \) was tested at increments of no smaller than one degree from -5° to +15° to ensure any advantage in \( L/D \) is captured.

![Figure 11. Effect of t/c on (a) drag and (b) maximum lift. (Simons, 1994 pp. 49-50, Fig. 4.11 and 4.13)](image)

D. Aerofoil Thickness to Chord Ratio

The aerofoil thickness to chord ratio (t/c) has a direct effect on the drag, maximum lift, stall characteristics and structural weight of the wing. As seen in Figure 11 (a), the drag increases with thickness due to the increased separation caused by the extra thickness. By contrast, as seen in Figure 11 (b), the effects of the thickness ratio on the maximum lift and stall characteristics of the aerofoil is primarily due to its effect on its nose shape, whose radius increases with thickness. This generally points towards an increase in maximum lift until about \( t/c=0.15 \) after which it peaks before dropping due to separation effects. Hence, particularly for low \( Re \) flight, most MAV wings apply the least amount of thickness possible, and generally employ a flat plate wing, as it also reduces the weight. The advantages of these aerofoils over wings with thickness are evident at the low \( Re \) region of MAV flight where small increase in the thickness of the wing can cause a significant
increase in separation early in the airflow, increasing drag and reducing lift (Abdulrahim, et al., 2002; Santhanakrishnan, et al., 2005). Hence, thin, undercambered wings allow the flow to remain attached on most of the wing, achieving higher $L/D$ ratios for thick wings. However, the Black Widow did show great success with a thickness ratio of 8%. This is generally perceived as the appropriate aerofoil design, as it enables the possibility of integrating the component internally and out of the airflow as well as keeping the flow attached at higher $\alpha$. Hence, the aerodynamic effects of employing a thicker aerofoil are perceived to provide little advantage relative to the separation problem and the weight penalty imposed by using this thicker aerofoil.

E. Wing Taper Ratio

Ideally, one would expect that the wing will carry the load uniformly from tip to root. However, this is only true for an elliptical wing where the effective $\alpha$ across the wingspan is constant and hence, the $C_{L,max}$ is reached everywhere simultaneously (see Appendix C for more detail). These results can be achieved however by introducing by tapering the wing.

In poorly designed wings, sections of the planform will provide effectively no lift and therefore become dead weight and overload the rest of the wing, and this effect is experienced in rectangular wing, whilst strongly tapered wings will experience the converse. For highly tapered wings, wing tips become prone to stalling due to the no tip chord length. Downwash on the wing causes an increase the effective $\alpha$ towards the tips, which for highly tapered wings with smaller wing area, easily overloads the wing tips and will stall virtually instantaneously. The advantages however of highly tapered wings are that although they are both dangerous and inefficient, they do provide a lighter wing without a reduction in strength due to its thick root.

Generally in aviation design, due to the expensive nature of producing and designing elliptical wings, and in particular the tooling costs, in order to achieve the ideal properties of an elliptical wing, segmented tapering is employed across the span of the wing, and if necessary, the tip shape is curved. However, again due to lift requirements conflicting with the dimensional limitations of MAV wings, taper isn’t generally pursued in the design. Alternatively, slightly elliptical design is generally employed due to the limited wing area available, and hence the study of the effects of wing taper has become a key component that requires analysis.

F. Other Considerations

One consideration of particular interest in the optimization analysis of the MAV wing is the surface roughness of the aerofoil. As seen in Figure 1, there is a suggestion that control issues seen on smooth aerofoils at low $Re$ may be negated by using a rougher surface to promote turbulence through the introduction of regular perturbations which introduce large disturbances into the flow to bypass transition. However, this brought about a reduction of lift which was found to be directly proportional to the height of the protuberances (Santhanakrishnan, et al., 2005). It must be noted however, as seen from Figure 12, that these protuberances promote early transition and hence delay separation similar to the controlling and smoothing effects of feathers on the flow around the birds’ wings (Chklovski, 2010). There are several noted difficulties in analysing these effects (Santhanakrishnan, et al., 2005), and hence for the undertaken study, the simplicity of the smooth surface was chosen as it would provide clearer results, whereas features such as the perturbations may be implemented in the future once the optimized wing is identified.

Another aspect which must also be considered is the use of flexible wings, such as a membrane wing to facilitate passive shape adaption to delay stall by reducing the lag time of the wing to adapt to the oncoming flow, providing a marked thrust generation in flap testing over a rigid wing up for $Re$ values less than $10^5$ (Heathcote, et al., 2004). By adapting to the free stream condition through adaptive washout and decambering (Abdulrahim, et al., 2002), these wings prevent flow separation, enhancing $L/D$, a result likened to the effect achieved by some bird species by the use of coverts that act like self-activated flaps (Shyy, 1999). However, this does impose a slight reduction in the overall lift coefficient (Viieru, et al., 2006). The adoption of a membrane wing, would not only defeat the purpose of testing the effect of the thickness ratio, but would also introduce aeroelastic effects which must be accounted for and analysed. This is beyond the scope of this research. However, aeroelastic tailoring (Weisshaar, et al., 1998) is a possible issue that may require increased attention in later research in this field, particularly if a flexible membrane wing is employed.
III. Rapid Prototyping Using Computational Fluid Dynamics

Rapid Prototyping is the use of a group of techniques to quickly fabricate a scale model of a part or assembly using three-dimensional computer aided design (CAD) data. This is an advantageous developmental method for new technologies as it not only decreases developmental time and the chances of costly mistakes but also ensures an extended product lifetime, allowing corrections to a product to be made early in the process (Chua, et al., 2003). Rapid prototyping allows an increased number of variant product articles to be produced and analysed, quickly enabling the compounding of increased functionality into the product by adding complexity and observing the consequent results on the products performance.

A. Result Accuracy, Precision and Reliability

There are three key concepts which must be addressed when undergoing CFD testing to ensure a meaningful result, namely: result accuracy, precision and reliability. Result accuracy pertains to attending to the setup of the CFD environment, ensuring that it is tweaked to mimic reality. Hence, it is more concerned with ensuring that the correct physics and physical shape is being tested and the results are verified using experimental data from physical testing.

Result precision, however, is related to how well the CFD algorithms solve the physics of the environment. This is achieved through analysis of the results outputted by the CFD program and ensuring that the result converges. This is because CFD is an iterative process based on reducing the residuals of the approximations. A converged result is characterised in Figure 13, and shows a smooth drop in the residual plots of several orders of magnitude with little to no oscillations. The problem is considered solved when the solution is converged and generally is observed with an overall reduction in the residuals of several orders of magnitude. Convergence of the results, however, merely shows that the result is precise and not necessarily accurate.

Oscillations in the residual plots are disconcerting and suggest that the results are unstable, questioning the setup and is generally an indication of transient flows. A reduction of one order of magnitude indicates that the calculated residual for that variable found by the calculation of the Navier-Stokes equation by the algorithm has reduced by a factor of 10. Hence, a precise result is one where there is a significant reduction in the residuals. However, a reliable result is not necessarily an indication of an accurate result. Hence, some form of verification testing is required.

The final concept is result reliability which is determined by ensuring that result is not mesh dependent. This can be due to failings in the mesh setup and mesh density, particularly in the crucial regions of the flow at and around the wing profile where the properties vary.

A CFD solution is only truly acceptable if it is accurate, precise and reliable.

B. Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) uses the Navier-Stokes equations of fluid mechanics in order to compute any parameter of any fluid at any point in a given flow field (Glowinski, et al., 1992). CFD is a powerful engineering tool that is being used in a wide range of engineering applications where flow analysis is required. In aviation, the use of CFD has become the standard prototyping method in design due to the time and cost savings incurred (Versteeg, et al., 2007). CFD is also used in MAVs aerofoils in a limited sense (Abdulrahim, et al., 2002). Solvers of the time-discretized Reynolds-Averaged Navier-Stokes (RANS) equations are the predominate flow modelling method currently used, and is the solving method used in FLUENT, the solver used in this research. Solvers used RANS iteratively to solve the mean flow and the effects of turbulence, and this is the preferred flow modelling method of analysis as it requires a relatively low amount of computing power to provide accurate results (Versteeg, et al., 2007).

The predominant issue in the use of CFD is ensuring the results are true to the reality they are attempting to model, particularly when implementing the turbulence model, which if implemented incorrectly will converge to an incorrect result. Furthermore, there is uncertainty in the CFD solution caused by the discretization process applied in order to calculate the differentials, and the possibility of using unconverged results. This occurs because of the use of an ineffective grid or the solver terminating before convergence is achieved due to the minimum residual being set too high. Hence, this is avoided by observing the residual plots throughout the iteration process to ensure convergence is achieved before the solution is terminated.

Figure 13. Converged FLUENT residual plot.
Furthermore, the validity of the CFD results must be proven to ensure that the results achieved are in fact indicative of the real solution. Some form of comparison of the results to experimental data should therefore be conducted. One method of doing so is dictated as follows:

1. Examine iterative convergence;
2. Examine consistency;
3. Examine grid convergence;
4. Examine temporal convergence;
5. Compare CFD results to experimental data; and
6. Examine any model uncertainties.

This methodology was used to confirm the obtained results as further research is undertaken.

IV. Project Overview

A. Project Intent

The purpose of this project was to undertake the testing required to design an MAV wing and to optimize and refine the wing with respect to its aerodynamic efficiency. This was achieved by analysing the effects of altering several geometric parameters of the wing, in particular $\alpha$ and $\phi$. Also, limited analysis was undertaken to examine the effects of altering these parameters in order to optimize the aerodynamic efficiency on the overall functionality of the wing.

For the purposes of this project, the aerodynamic efficiency was defined as the ratio of the lift force generated by the wing and the total drag experienced by it ($L/D$). Although achieving the maximum possible $L/D$ would be the ideal design, the optimized wing was not defined as being merely as such. It was important to ensure that the design was also realistic and producible. Hence, the practicality and the manufacturing process requirements of the design, along with $L/D$ achieved, were the three main objectives of the optimal wing (See Appendix D and E).

It is hoped that the work undertaken can be used as the initial phase of developing a school MAV for further research and development.

B. Methodology

The planned process that was undertaken to achieve the project intent involves several stages and follows a similar approach as to that used in similar projects (Kell, 2009). The project work has been segmented into the following six task components.

1. Project Management Planning:

The initial step undertaken in this project was the implementation of a project management plan (PMP) which details a succinct outline of the project expectations based on the client brief negotiated between the researcher as project manager and the project supervisors as clients. The plan details a comprehensive list of tasks, expected work breakdown, expected subtasks required to be completed to complete the task and the associated deadlines to ensure the successful completion of the project in a timely manner. The PMP was reviewed on a monthly basis and corrected as necessary and includes a work breakdown structure, project schedule and a milestone chart. The PMP is attached as Appendix D. Furthermore, an in-depth study of all the expected influential parameters to this study in a conceptual design development and comparison task was undertaken (provided in Appendix E). This enabled the determination of the key factors in this study and their interrelationships and dependencies, allowing a broader understanding of the task at hand.

2. Literature Review

A literature review was conducted, to document and analyse previous relevant research in the field – in order to ensure that the researcher of this project achieved a sound understanding of the type of research has previously been undertaken in the field. The literature review also ensured that the researcher was able to critically analyse it and take in an understanding of the possible expected results and issues that might be incurred. Additionally, in order for the researcher to understand the defining features of the research to be undertaken that distinctly set this project apart from previous studies. Furthermore, this research enables a greater understanding of the fundamentals of low $Re$ aerodynamics and MAV design, and determining the appropriate prototyping method.

3. CAD Modelling

All CAD models were constructed by the author using a macro in Excel to import data points into CATIA. A simple parametric control wing was constructed with systematic stepwise alteration of one parameter at a time, to produce the required test articles.

4. CFD Solution

The produced CAD models were meshed through the ANSYS Workbench mesh tool before being tested in FLUENT to convergence. The study concentrated specifically on the wing, and assumed the analysis as if it were a flying wing design, ignoring all issues of rotation and flow interaction with the propwash of the
propulsion propeller (most probable method) as these elements have not been defined or specified in either shape, size or type. It must be noted, however, that these effects have been observed to drop the $L/D$ of the UF MAV by two-thirds (Abdulrahim, et al., 2002). Furthermore, this allowed a simplification of the analysis. A mesh study was conducted on the control wing. This produced an appropriate mesh which converged precisely and provided reliable results which were used for further testing. This step required familiarity with FLUENT and the ANSYS Workbench as well as an understanding the methodology required to construct the requisite physics to produce the correct CFD framework that provided the most correct results for analysis. This knowledge was acquired by completing the course Computational Fluid Dynamics in first semester and undertaking the extra tutorial programs in allocated thesis work times.

5. CFD Verification

It was necessary to verify these results, at least for a select few designs, in order to ensure confidence in the obtained results. This verification must be done so that the CFD problem is exactly the same as the physical test (Martinez 2004). It was hoped testing on a manufactured model would take place using the ADFA low speed wind tunnel, but due to time constraints, this step was not realised to its full potential. The results were partially verified against the calculated results from Prandtl’s Lifting Line Theory (PLLT) which is an aerodynamic theory mathematical solution derived from experimental modelling.

6. Design Evaluation

From the flow field visualised by the verified CFD results, it was then possible to analyse the results with confidence and hence determine the most aerodynamically efficient value of the geometric parameters of the aerofoil.

V. Initial CFD Setup

A. Initial Setup Overview

This chapter outlines the process undertaken to setup the initial CFD testing framework to ensure a reliable testing environment. This process enabled the use of the rapid prototyping technique of the appropriate wing variants without the requirement to verify each individual result. This chapter also discusses the method used to construct and import the wing shapes into the software. The implications and considerations undertaken in order to construct the applicable mesh system and set up the physics required in order to analyse the desired wing geometries in the CFD environment are also discussed.

B. Wing Generation

The initial setback that arose in the testing process was the method required for generating an accurate wing shape that is compatible with the CFD software or programs which are capable to import geometries into the software such as CATIA. The predominant problem encountered was that there was no drawing method available in the software capable of either drawing the chosen NACA 0008 airfoil based on its cross-sectional formula nor was there a software such as CATIA shape that is compatible with the CFD software that will enable the automatic manipulation of the airfoil shape by altering parameter values.

In order to overcome these issues, a Microsoft Excel program named MAV Wing Template was created to set up the wing and a macro was used to import the wing into CATIA which drew and filled the wing shape and created a usable model that was imported into the ANSYS software.

The Excel program is based on 200 (initially 20) $t/c$ data points of the NACA 0008 airfoil that were sourced from the NACA 4 Digits Series Profile Generator (Trapp et al., 2010) and verified using NACA’s summary equations for its airfoil data (Abbott et al., 1945). These data points are used to produce the airfoils sections at three wing half span stations in the form of X, Y and Z coordinates. The origin was defined as being the tip of the leading edge at the wing centre. The system defined positive X direction as the flow direction, positive Y as MAV starboard and positive Z as up. The program manipulated the wing aerofoil profile through the use of geometric correlations (see Appendix F for simplified program calculations) taking into account the geometrical characteristics being analysed, namely, the $AR$, $\lambda$, $\phi$ (and twist location) and $\delta$. The program also took into account the MAV dimensional constraints, automatically setting the wingspan to 150 mm.

For the initial setup, the test wing that was used was the $0^\circ\phi$ and $0^\circ\delta$ wing condition as pictured in Figure 14. The setup dictated that $\delta$ would be taken from the leading edge. For the testing regime analysed, the $AR$ was set at 2.0 to increase efficiency and $\lambda$ was set to 0.5 to approximately mimic the elliptical wing distribution.

The outputted 600 points from the template program were then entered into a separate spreadsheet that contained the embedded macro which was sourced from the CATIA User Companion. The macro constructed

![Figure 14. Test wing shape](image)
the wing by importing the points into a CATIA part and forming splines for each of the wingspan sections before lofting a surface between the splines, giving the wing shape.

Once imported into CATIA and checked, the wing was then closed using the close surface tool before being saved as a step file (*.stp), the supported file type for ANSYS suite.

C. Computational Fluid Dynamics Analysis using ANSYS 12 Workbench

The main program used in the CFD study was the ANSYS Suite. A workbench of three components was created. The ANSYS Workbench integrates FLUENT (the flow modelling software) with other main ANSYS tools. Of particular note are the two other programs used, namely, the ANSYS Mechanical Modeller, which modifies and creates the testing domain geometry, and the ANSYS Mesher which creates the complex meshes required.

The setup of the CFD environment in ANSYS defines several important factors that significantly influence the accuracy of the solution obtained. These include the fluid geometry, the mesh sizings and inflations as well as the boundary conditions and fluid properties. These shall be discussed below.

1. Mechanical Modeller

The Mechanical Modeller component of the ANSYS workbench creates the geometrical model of the test domain being studied. There are several factors that require consideration when determining the environment geometry, in particular size, shape and simplicity. As the essence of the design is to replicate the ideal testing conditions of an infinitely large wind tunnel which produces perfectly laminar flow, the test domain must be large enough to ensure that there are no residual effects captured due to interference effects such as the interaction of the wing with the boundary walls of the domain. Negating these problems are not only dealt with when setting up the boundary conditions at the test environment walls but also by determining the size of the test domain. Although the larger the size of the environment the less the interaction effects are observed, increasing the size implements an increased penalty arising from the excess redundant calculations required during the CFD analysis for the domain space which is not affected by the presence of the wing. According to Holloran et al. (1999), who tested the effects of wing in ground effect in 2D, it is suggested that in order to ensure the test captures the disturbances due to the wing entirely, the wing should be separated from the wall boundaries by a minimum of two wingspans, a result which was also shown to be true in the 3D condition (Appleton et al., 2010). Therefore, to ensure that the effects of the presence of the wing do not extend beyond or are influenced by the domain boundary whilst also incorporating a simplistic model which is computationally conservative, it was decided that the test environment would be defined as a one metre cube using the same axis convention as specified previously. The wing was imported from CATIA step file into the geometry and placed with its origin at the centre test environment.

Upon importing the step file into the mechanical model, the wing is then created in the environment domain and is observed as consisting of several new parts/bodies – these being the desired filled wing profile and other unwanted artefacts from the step file such as the lofted surfaces which defined the wing geometry in CATIA. As FLUENT can only analyse one part (made out of several named surfaces), the wing part is subtracted from the original cubic metre environment using a Boolean operation, creating one part. The remaining artefact parts were suppressed from any further analysis. Furthermore, a Body Operation was created for the imported wing, rotating it about the Y-axis enabling the control of \( \alpha \). A nominal \( \alpha \) of \( 2^{\circ} \) was decided as the base testing angle, since by using the smooth symmetrical NACA 0008 wing aerofoil, the lift on the wing is theoretically zero (on a \( 0^{\circ} \) wing).

Hence, by introducing an \( \alpha \) and introducing an asymmetry relative to the airflow, lift can be produced.

2. Mesher

The Mesher component in ANSYS creates the mesh throughout the fluid, determining the points at which the RANS equations are calculated throughout the domain. In CFD analysis, in conjunction with setting up the boundary conditions, setting the mesh is the most vital aspect of the process, as it determines the reliability

![Figure 15. Setup of test domain mesh.](Image)

Inset: close-up detail of mesh around wing.
and precision of the results. Simply increasing the mesh density throughout the test fluid geometry will increase the result reliability. However, by decreasing the cell dimension by half this will not only increase the number of calculations required to reach convergence but will also increase time to convergence of the 3D solution by a factor of eight. Therefore, not only were the increases in densities regionally selective but also a compromise of accuracy and time to convergence is chosen in order for the rapid prototyping testing to be practical. It was therefore decided that an unstructured tetrahedral mesh would be used throughout the domain. The mesh was designed so that it was sensitive to the proximity and curvature around the wing and was denser in those regions. The use of the unstructured mesh provided the advantage of an easier set up and one that generated quickly relative to a structured mesh. The unstructured option also provided the ability to create a mesh which expanded with larger cells towards the edges of the test environment where the changes are expected to be minimal while the changes around the wing are captured. Furthermore, as the testing will also include a turbulence model, a boundary layer is expected to form over the surface of the wing and hence, in order to capture the boundary layer along the zero slip surface, a dense rectangular structured inflation layer was created along the wing.

The determination of the final values of the initial tetrahedral mesh size, grid refinement studies were conducted on the mesh and are detailed in Section D of this chapter. It must be noted however, that the expertise in the use of the program was minimal at the commencement of the project. As discussed later in this report, as the project continued, there were adjustments required to improve the mesh.

3. FLUENT

FLUENT is the ANSYS product of choice in modelling, testing and analysing the fluid flow in complex geometries and hence, was sufficiently capable of analysing the uninterrupted clean flow over the MAV wing accurately. However, due to the complex nature of the program, many considerations had to be made in deciding the composition of the final CFD setup.

The first aspect of the problem setup that had to be considered was the turbulence modelling of the flow. Although it was accepted that turbulence effects would be present in the solution, and in particular at the wingtip, it was assumed that the result would be steady state and that no transient effects would be present. This assumption was based on the knowledge that the inflow was set to be uniform and steady over a fixed wing at very low speeds, and hence the uniformity of the flow was expected to continue. This result is extremely favourable for the rapid prototyping analysis as it reduces the problem complexity and the time required for the analysis. Although some literature suggested for low Re flows the standard k-ω model is to be used (Chilka et al., 2010), the k-ε model provided the advantage of being the simplest and most widely validated turbulence model. The k-ε model is the simplest of the two equation models and transports the two named variables, the turbulent kinetic energy, k, which determines the energy of the turbulence, and the turbulence dissipation, ε, which determines the scale of the turbulence. Considering the simplicity of the boundary conditions discussed below, the realisable k-ε turbulence model was determined to be the most appropriate for the analysis. Also, the consideration had to be made that it was expected that a boundary layer with a possibility of separation would be present. However, it was unlikely that there would be sudden changes in the flow direction and hence the turbulence could adjust to the local conditions (FLUENT, 2006). The robustness of the solution was therefore assured despite its poor rotating flow analysis (Versteeg, et al., 2007).

The turbulence specification method used in defining the model was the intensity and viscosity ratio. This method required the input of two variables when determining the turbulence at the inlet and outflow of the test region. These inputs are the turbulence intensity (I) (Equation 2) and the turbulent viscosity ratio (β) (Equation 3) (FLUENT, 2006).

\[ I = \frac{u'}{u} \quad (2) \]

\[ \beta = \frac{\nu_t}{\nu} \quad (3) \]

Although as part of the FLUENT solution the value for both of these variables at each cell is calculated, there are limiting values for these which are defined in the boundary conditions, and which help to determine the accuracy of the results. There are several guidelines when determining these limiting values. The FLUENT User guide suggests that for low Re flows it is expected to be a medium-turbulent case, and the typical I value for this type of case is expected to be between 1% and 5% while the value for β should lie in between 1 and 10 (FLUENT, 2006). Hence, for the setup, a limiting value of 5% and 10 respectively was chosen, with the results inspected as part of the analysis.

The movement of the wing in air in the boundary conditions setup, similar to a wind tunnel, was modelled with the wing was kept stationary, while the front face was defined as a velocity inlet with a uniform absolute velocity magnitude of 15 m/s – which is the maximum velocity at which the an MAV is expected to travel. This set up the Re of the wing in the order of 75,000 – which places the wing just below the sudden rise in \( \frac{L}{D_{\text{Max}}} \) expected at low Re numbers as predicted in Figure 1.

Thesis Report 2010, SEIT, UNSW@ADFA
In order to obtain accurate results, the boundary conditions applied must be closely examined. The interpretation of the fluid dynamics in MAV flight is that it should theoretically be assumed to be incompressible due to the low speeds and is in the low Re region. (Abdulrahim, et al., 2002) Viscosity is accounted for indirectly by imposing the Kutta condition on the trailing edge and including the influence of the vortical structure on the leading edge (Zwikowski, 2002). This condition fixes a fluid stagnation point at the trailing edge, ensuring flow above and below a flat wing fuse smoothly at the sharp trailing edge.

Reducing the shear effects at the boundary walls of the test environment was achieved by modelling the side, top and bottom faces of the domain as moving walls with a uniform absolute velocity value of 15 m/s and zero roughness, hence mimicking infinite air boundaries on the freestream flow. Other boundary conditions chosen were the basic specification of the fluid being the standard FLUENT air model and defining the back wall as an outflow boundary with no backflow, i.e. 0 Pa gauge pressure.

The final aspect of the setup is determining the solution methods. During the initial startup FLUENT was setup to work in 3D and use double precision to accurately capture the numerical results. Furthermore, the spatial discretization methods in the solution method controls were setup as second order upwind for turbulence and momentum models which is the most accurate method available and is the expected standard, while the gradient was calculated using the Least Squares Cell Based method. The pressure model was also initially setup as second order. However, due to convergence issues the model was returned to first order, and the pressure-velocity coupling method was left in the SIMPLE method.

D. Mesh Refinement Study

1. Wall y+ test

As stated previously, upon setting up the FLUENT for a test case, testing must be carried out to ensure that the mesh provides realistic results and that the results are not mesh dependent. For turbulent flows there are two main tests which are undertaken. The first test is determining the minimum inflation layers required to sufficiently capture the boundary layer over the wing. The test is done by comparing the maximum value of the non-dimensional wall distance y+ for the flow over the wing (Equation 4) which gives an indication of how well the boundary layer is being captured.

\[
y^+ = \frac{u_y}{u} \quad (4)
\]

\[
u_+ = \sqrt{\frac{\tau_w}{\rho}} \quad (5)
\]

Where \( u_+ \) is calculated using Equation 5.

Ideally for the flow over the wing the wall y+ values should be in the order of one. However, higher values are acceptable as it is well inside the viscous sub-layer (FLUENT, 2006), with the smaller the value the more accurately the solution will capture the boundary layer. As shown in Table 3, it is evident that the y+ value decreases with more inflation layers. However, by increasing the number of inflation layers there is an inherent increase in the number of mesh cells and therefore an increase in the computational time.

<table>
<thead>
<tr>
<th>Inflation Layers</th>
<th>Wall Y+</th>
<th>Number of Mesh Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>76.4</td>
<td>398112</td>
</tr>
<tr>
<td>5</td>
<td>11.0</td>
<td>515676</td>
</tr>
<tr>
<td>7</td>
<td>8.59</td>
<td>542304</td>
</tr>
<tr>
<td>10</td>
<td>5.17</td>
<td>573428</td>
</tr>
</tbody>
</table>

Table 3. Analysis of the effect of number of inflation layers on wall y+.

Hence, based on the results, seven (7) inflation layers were chosen for the initial testing as it was seen as the most practical value to achieve the results required. This configuration resulted in a y+ plot over the wing exhibited in Figure 17 (a). The wing surface predominately has a y+ result below five, particularly at the wing tips and edges. The larger y+ values are a smaller portion of the wing at the centre of the wing surface area.

The outcome of addition of these inflation layers is reflected in the final solution of the flow. Figure 17 (b) depicts the velocity (u) magnitude vectors, with the seven inflation layers at the wing surface capturing a distinct boundary layer.

Figure 16. Plot of inflation layer analysis based on maximum wall y+ results.
2. Grid Refinement Study

The second test that was undertaken in the grid refinement study is a regression study to determine the most appropriate mesh spacing that would provide an acceptable compromise between accuracy and computational time. This is required as it is not practical to analyse an extensively dense mesh due to the limited computational resources available. Furthermore, the mesh must be practical for the rapid prototyping analysis being conducted which requires a large amount of tests with small variations in the geometry. Hence, the computational time for the testing mesh must be acceptable so as not to detract from the advantages of the technique when compared to physical testing.

Table 4. Analysis of the effect of minimum sizing on lift and drag results to determine result reliability.

<table>
<thead>
<tr>
<th>Min Size (mm)</th>
<th>Cell Number N</th>
<th>Lift (N)</th>
<th>Drag (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Very Course</td>
<td>1.727</td>
<td>365163</td>
<td>0.1404</td>
</tr>
<tr>
<td>Course</td>
<td>0.863</td>
<td>453736</td>
<td>0.1398</td>
</tr>
<tr>
<td>Normal</td>
<td>0.432</td>
<td>542304</td>
<td>0.1396</td>
</tr>
<tr>
<td>Fine</td>
<td>0.216</td>
<td>893652</td>
<td>0.1391</td>
</tr>
</tbody>
</table>

In order to ensure the chosen mesh spacing (normal) results in a suitably low amount of discretization error, four mesh spacings were evaluated by varying the size of the minimum spacing (Table 4). An analysis of the two main outputted results required in this study, namely the wing lift and drag forces, was undertaken once the solution converged for all four meshes through the use of a Richardson extrapolation.

A Richardson extrapolation provides an indication of the expected level of uncertainty in the reliability of the results. This method extrapolates the theoretical result of an infinite cell mesh by utilising a relationship between the results computed from the different meshes. The extrapolation uses successive results of a given variable $S_A$ and $S_B$, for two different meshes with cell densities $N_A$ and $N_B$, having a bias towards the denser meshes. The solution of the extrapolation, $S_\infty$, is the expected result which is associated with a grid with an infinitely small grid-spacing.

$$S_\infty = \frac{-1}{1-(N_A/N_B)^p}S_A + \frac{(N_A/N_B)^p-1}{(N_A/N_B)^p-1}S_B$$  \hspace{1cm} (6)

Although the momentum methods used in the CFD analysis were second order, the use of the first order pressure model would reduce the order of convergence. Hence, by initially starting with an assumption of $P = 2$, a regression analysis was conducted to check for the
order of convergence of the code. The natural log of the uncertainty in the results (difference between the calculated and the extrapolated values) \( \ln(\epsilon) \) was plotted against the log of the number of grid points \( \ln(N) \), and the trend line gradient provided the order of convergence of the results (Figure 18). Both the lift and drag analysis showed good convergence with 1.93 and 1.84 respectively and gave promising results (see Figure 18 and Table 5). Using these final results, the uncertainty in the results from the chosen normal mesh and the extrapolated lift and drag forces was given at 0.51% and 0.81%, giving a final uncertainty of the mesh for the \( L/D \) ratio of 1.03%.

There is also a level of uncertainty in the results due to the numerical truncation of the results by FLUENT to at least six significant figures. However, this was considered to be too exhaustive of an answer and such a truncation error would be negligible as the results were taken to have meaning to up to 4 significant figures as this was deemed a more than ample realistic result.

Despite the relatively low uncertainty in the results, it is noted that the mesh was designed to be sparse for the purpose of reducing the computational time for the rapid prototyping analysis. However, this does put the system at risk of being compromised if the design varies excessively, an effect which was experienced with this mesh in later analysis of the \( \Lambda \) testing as discussed below.

Although, only the wing lift and drag data were required for the analysis, several other plots and data results were captured for each test and used to analyse if there are any anomalies in the results. A sample collection for one test case is given in Appendix H.

![Richardson Extrapolation for Lift Value](image1.png)  
(a)  
![Richardson Extrapolation for Drag Value](image2.png)  
(b)  

**Figure 18.** Final Richard extrapolation plots after order of convergence was determined for (a) the wing lift and (b) the wing drag of the test wing.

<table>
<thead>
<tr>
<th>Extrapolated Result</th>
<th>Lift (N)</th>
<th>Drag (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chosen Mesh Result</td>
<td>0.1389 N</td>
<td>0.0422 N</td>
</tr>
<tr>
<td>Difference</td>
<td>-0.0007 N</td>
<td>0.0003 N</td>
</tr>
<tr>
<td>Uncertainty</td>
<td>0.51%</td>
<td>0.89%</td>
</tr>
<tr>
<td>Order of Convergence</td>
<td>1.93</td>
<td>1.84</td>
</tr>
</tbody>
</table>

**Table 5.** Results of Richardson Extrapolation.

E. Characteristics of Mesh Reliability Results

The overall convergence plot for the agreed mesh setup produced a distinct curve for the scaled residuals. After the initial spike of several orders of magnitude in the first 50 iterations, the residual curves subside and reduce consistently for 700 iterations before going through a sudden drop of three to four orders of magnitude over about 200 iterations, converging at about 1,000 iterations with minimal reduction in the residuals beyond that point. Hence, an ideal a run time of 1,200 iterations ensured that all tests would converge acceptably within the testing scheme. However, it was decided that 1,600 iterations was an ideal number of iterations for extensive studies in the computer lab. This accounted for about 2.5 hours of computational time, and enabled the user to individually set up each of the 24 computers in the Lab and return to find the first test finished and ready for data collection.
The final convergence plot at 1,600 iterations gave a reduction in scaled residuals of at least five orders of magnitude \([O(10^{-5})]\) for both the transported turbulence model variables \(k\) and \(\varepsilon\), while the momentum model reduced by a minimum of \(O(10^{-8})\) for each of the velocities in the three axis while the continuity equation reduced to a order of magnitude \(O(10^{-5})\). The lack of oscillations confirms that there are no transient effects in the flow as expected.

This result was deemed appropriate due to the desired nature of the mesh to be only used to produce trend results and not necessarily for providing precise individual results.

F. Verifying Results using Prandtl's Lifting Line Theory

Validation of the results is a vital confirmation of the results. A check was conducted using the Prandtl's lifting line theory (PLL) to calculate the wing lift and drag coefficients and hence the subsequent forces on the wing. This calculation takes into account the wing geometry for thin wings and is accurate for wings with maximum thicknesses less than 0.12c. The theory nests several horseshoe vortices that add circulation to the lifeline of the wing where the vortices interact. Horseshoe vortices comprise of bound (fixed) vortices, which are at the leading and tip edges of the wing segment and a free vortex at the segment trailing edge tip. Hence, by calculating certain parameters at several wingspan stations a angular displacement \(\theta_0\), from the wing root it is possible to solve for the coefficients \(A_n\) as given in the monoplane equation (Equation 7) using a simple Excel program and MATLAB (See Appendix G).

For the analysis 5 horseshoe vortices were applied to the analysis of a half wingspan, with the final result doubled to account for the full wing. Equation 7 is a variant of Prandtl's which takes into account wing twist \(\phi(\theta_0)\) (Katz et al., 2001).

\[
\mu[\alpha(\theta_0) - \phi(\theta_0)] \sin(\theta_0) = \sum_{n=1}^{N} A_n \sin(n\theta_0) [\mu n + \sin(\theta_0)]
\]

where the parameter \(\mu\) is calculated using Equation 8.

\[
\mu = \frac{C(\theta_0) \sin(\theta_0)}{\pi b}
\]

with the final Lift \((L)\) and Drag \((D)\) results being calculated using Equations 9 and 10.

\[
L = 0.5pu^2A_1\pi b^2
\]

\[
D = 0.5pu^2\pi b^2 A_2^2[1 + \sum_{n=2}^{N} n(n/A_2)^2] + D_0
\]

PLLTT however has certain limitations. Firstly the calculations are based on an elliptical wing. However, as the \(TR\) chosen was 0.5, the stall characteristics of the wing should be close to that of an elliptical wing, although \(x = 0.4\) is more ideal. Furthermore, the classical theory is poor for predictions for wings with an \(AR\) of less than 4 due to its predictions of the 3D lift curve slope \((\alpha(\theta_0))\) being too high. Similarly, the classical theory does not take into account the effect of swept wings. Hence for this analysis \(\alpha(\theta_0)\) was calculated using Equation 11 which uses both the Helmbold and Kucheman modifications for low \(AR\) and \(A\) respectively (Katz et al., 2001).

\[
\alpha(\theta_0) = \frac{a_0(\theta_0) \cos(A_{rad})}{1 + a_0(\theta_0) \cos(A_{rad})}
\]

\(a(\theta_0)\) was calculated to be 4.655 \pm 0.003 which is an uncertainty of 2.7%. This result was reached by taking the 2D lift curve slopes \((a_0)\) for the NACA 0006 and the NACA 0009 summary aerofoil data and a result for the NACA 0008 was extrapolated. This result gave an uncertainty for the lift and drag results of 3.9%.

VI. Initial Testing Results

A. Initial Testing Overview

An initial testing schedule based on the discussed range of variation in the two wing geometric aspects being explored. Based on the literature review (Simons, 1994) \(\varphi\) was initially tested at increments of 1° between -5°\(\varphi\) and 5°\(\varphi\). However contrary to the literature, upon analysing the data it was found that by increasing \(\varphi\) beyond 5°\(\varphi\) the wing \(L/D\) continued to increase and hence, the testing regime was extended to +15°\(\varphi\). Similarly, the test regime for \(A\) was increased from the original testing between 30°\(A\) forward and 30°\(A\) back at increments of 5° to include maximum back of 55°\(A\), at which state the maximum dimension of 150 mm was reached longitudinally. Lift was defined as all force in the Z direction and is independent of \(\alpha\) or \(\varphi\), similarly the drag was defined as the force in the X direction.

18

Thesis Report 2010, SEIT, UNSW@ADFA
As discussed in the previous chapter, the mesh was setup to determine trends and there was a possibility for it to break down. This was justified as the aim of the initial testing was to produce a quick assessment by conducting a large number of tests to provide a decisive indication of the trends which are to be expected by varying the different parameters. In order to justify the observed trends, each of the two tested parameters was tested at three consecutive increments of the other parameter. Analysis of the PLLT calculated results predicted that the variations between consecutive increments of $\Lambda$ for a given $\phi$ angle are expected to be virtually negligible for all $\phi$ values. PLLT also suggested that the difference between consecutive increments of $\phi$ would be a distinct offset which is constant for all sweeps. Hence, by increasing the range of available test data, it became possible, as did the capability to compare and analyse the results and determine whether and where the setup may require to be adjusted.

B. Twist Testing

1. Initial Twist Results

Initial $\phi$ results are shown in Table 6 and Figure 19. The results appear to conform extremely well. The results show a great affinity between the sequential curves of different $\Lambda$, with an average standard deviation of 0.22% between the mean curve and the separate results. There is a distinct trend observed in the results, with an increase in $L/D$ until a $\phi$ of about 11° before dropping. The increase is not uniform, but is a decreasing increase with each extra degree of $\phi$ before reaching the maximum before decreasing at an increased rate. As the reduced advantage with each increased degree of twist, confidence in this result is very important, particularly in the region of maximum $L/D$.

An analysis of the $L/D$ gradient as shown in Figure 21 shows the scatter of the gradient change conforming to an almost linear plot particularly for $\phi > 0°$. The results also show the standard deviation between the three curves decreases substantially with increased $\phi$ particularly in the high $L/D$ region, increasing the confidence in this result.

### Table 6. Tabulated wing twist variation data

<table>
<thead>
<tr>
<th>Twist Angle</th>
<th>$L/D_{mean}$</th>
<th>$L/D_{mean} - L/D_{max}$</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>-5</td>
<td>0.3751</td>
<td>6.9520</td>
<td>-1853.63%</td>
</tr>
<tr>
<td>-4</td>
<td>0.3235</td>
<td>6.2535</td>
<td>-1933.33%</td>
</tr>
<tr>
<td>-3</td>
<td>1.1283</td>
<td>5.4486</td>
<td>-482.90%</td>
</tr>
<tr>
<td>-2</td>
<td>1.7393</td>
<td>4.8376</td>
<td>-278.14%</td>
</tr>
<tr>
<td>-1</td>
<td>2.4612</td>
<td>4.1157</td>
<td>-167.23%</td>
</tr>
<tr>
<td>0</td>
<td>2.9504</td>
<td>3.6265</td>
<td>-122.92%</td>
</tr>
<tr>
<td>1</td>
<td>3.7629</td>
<td>2.8140</td>
<td>-74.78%</td>
</tr>
<tr>
<td>2</td>
<td>4.3423</td>
<td>2.2346</td>
<td>-51.46%</td>
</tr>
<tr>
<td>3</td>
<td>4.7592</td>
<td>1.8177</td>
<td>-38.19%</td>
</tr>
<tr>
<td>4</td>
<td>5.2177</td>
<td>1.3592</td>
<td>-26.05%</td>
</tr>
<tr>
<td>5</td>
<td>5.6212</td>
<td>0.9557</td>
<td>-17.00%</td>
</tr>
<tr>
<td>6</td>
<td>5.9131</td>
<td>0.6638</td>
<td>-11.23%</td>
</tr>
<tr>
<td>7</td>
<td>6.1548</td>
<td>0.4221</td>
<td>-6.86%</td>
</tr>
<tr>
<td>8</td>
<td>6.3479</td>
<td>0.2290</td>
<td>-3.61%</td>
</tr>
<tr>
<td>9</td>
<td>6.4806</td>
<td>0.0963</td>
<td>-1.49%</td>
</tr>
<tr>
<td>10</td>
<td>6.5516</td>
<td>0.0253</td>
<td>-0.39%</td>
</tr>
<tr>
<td>11</td>
<td>6.5769</td>
<td>0.0000</td>
<td>0.00%</td>
</tr>
<tr>
<td>12</td>
<td>6.5440</td>
<td>0.0329</td>
<td>-0.50%</td>
</tr>
<tr>
<td>13</td>
<td>6.5006</td>
<td>0.0764</td>
<td>-1.17%</td>
</tr>
<tr>
<td>14</td>
<td>6.3918</td>
<td>0.1851</td>
<td>-2.90%</td>
</tr>
<tr>
<td>15</td>
<td>6.2358</td>
<td>0.3411</td>
<td>-5.47%</td>
</tr>
</tbody>
</table>

### Figure 19. Effect of varying wing twist on L/D

**Figure 20. Pressure distribution around the 15° twist wing at (a) the wing root and (b) the wing tip.**
2. Analysis

The distinct peak in the results would be expected to be due to the wing outboard section stalling as seen in Figure 20. This is expected as maximum lift should theoretically be at an incidence angle of 13° according to the 2D lift curve slope of the NACA 0009 airfoil (Abbott, 1945). This translates to the 11°φ at the wing tip plus the α = 2° where the maximum L/D lies. However, upon further inspection of the separate lift and drag data as seen in Figure 22 (a), the results suggest that the lift remains to increase at a constant linear rate of 0.053 ± 0.008 N/°φ, despite the outboard stalling of the wing. However, this consistent rise in lift is attributed to the increased φ on the inner portions of the wing, causing the extra lift to be generated inboard. Hence the main reason for the loss in L/D after the 11°φ mark is due to the increase in drag at that point being so much greater than the increase in lift in that region (Figure 22 (b)).

3. Comparison to Prandtl's Lifting Line Theory and Laminar CFD Testing

Figures 23 (a)-(b) show the comparison of the separate CFD lift and drag turbulent results compared to the analytical results achieved using PLLT. Laminar tests similar to those described for the turbulent flow were also
conducted with the results used in the comparison figures. In the lift case there is virtually no difference in the CFD results. Furthermore, compared to PLLT, the CFD lift results are remarkably similar, being lower than the PLLT results by, on average, 0.060 ± 0.015 N. The two solutions are converging with higher φ angles, with the PLLT and the CFD results showing a gradient of 0.0307 and 0.0323 N°φ respectively. However, PLLT does suggest a lift at 0°φ of 0.203 N, 67% higher than the CFD results. Furthermore, in the region of interest the difference in results is 0.061 N (14.9%) at 7°φ and 0.031 N (4.8%) at 15°φ.

Figure 23. Comparison plots of the CFD data and the calculated PLLT results for the (a) wing lift force and (b) wing drag force when varying wing twist.

For the comparison of the drag results (Figure 22 (b)) there is a definite parabolic trend with increased φ in all the curves. However, unlike the lift data, the CFD laminar and turbulent flows do not correlate as well due to the turbulent flow causing the greater drag on the wing. There is greater excess drag in the turbulent model relative to the laminar results at lower φ, with the results appearing to converge at higher φ. It must be noted that the major difference between the theoretical and CFD results is due to the fact that the PLLT only calculate the wing induced drag (Di) and does not include the profile drag (D0). An estimate of the profile drag using the airfoil data (Abbott, 1945) gives CDo of 0.005 and hence, a D0 of 0.0075, which is completely negligible in its effect on the result, and therefore D0 was not included. D0 is expected to be a virtually constant offset and will have no effect on the overall result of the trend. However, it is noted that there will be minimal increases in D0 with increased φ due to the increased cross sectional area of the wing relative to the oncoming flow and the increased drag polar due to the increased CL. The comparison curves show that the curves are incredibly similar with the turbulent CFD plots with a difference of (D0 - 0.038) ± 0.0017 N between the two curves, suggesting that the results are reliable.

Although exhibiting similar trends, the consistency between the PLLT results and the laminar results are not as fine. Despite the difference, the high affinity between the two models in the trend does suggest that the results are acceptable particularly for the analysis undertaken. Furthermore, these results for the turbulent CFD model consistently give a more conservative L/D result than is shown in either the PLLT or the laminar case, as not only does the turbulent model show a lower lift when compared to the theoretical results, but also shows a higher CD when compared to the laminar case.

4. Conclusion

When choosing an appropriate wing design it is important to ensure that the chosen design is of a robust nature. In the current environment, it is most optimal to use a wing with 11°φ. However, between 9°φ to 13°φ, the results vary by less than 1.5% from the maximum. Therefore, it is not necessarily the most ideal, as discussed to earlier. In reality, the conditions to which this wing will be subjected are not necessarily uniform, and there is a great likelihood that the wing will experience gusts and other anomalies in the airflow which will cause an imbalance in the lift generation across the wing causing pitch up effects, and hence if the wing is at the optimum φ = 11° for the design α = 2°, a pitch up of 3° equates to a 2.9% loss in lift. This effect is more apparent in the laminar condition where the maximum peak is greatly pronounced (Figure 25) as the drag, although always lower than in the turbulent condition, increases at a greater rate. Furthermore, the peak is also
achieved at $7^\circ \phi$, with the reduction of 8.5% in $L/D$ between $7^\circ \phi$ and the optimum $\phi$ in the turbulent condition of $11^\circ \phi$.

Furthermore, one of the fundamental design requirements is practicality and simplicity, excessive $\phi$ does pose structural integrity risks particularly for balsa wood models as the $\phi$ will be with the grain and hence weakening the wing, particularly considering the thin airfoil (4 mm maximum thickness at the tip) relative to the 13.6 mm height at $11^\circ \phi$. Whilst by using a wing with just $9^\circ \phi$ the height reduces to only 9.9 mm.

The contrary argument is that, as shown in Figure 24, the maximum $L/D$ for the wing occurs at slightly higher $\phi$ with reduced speed. Similar to the results for the 15 m/s velocity, the other results were also verified using PLLT. Hence, even by using an $11^\circ \phi$, the resultant wing will have an $L/D$ 3-4% lower than that it can achieve $L/D$. However, as can be seen from the graphs the peaks become less prominent with lower speeds and hence the pitch effects due to environmental anomalies would impact the overall performance of the wing less as it would be more stable.

Taking these design aspects into consideration it was decided that the most appropriate design would be to use the $9^\circ \phi$ twist wing which although is a reduction of 1.5% in the maximum $L/D$, is a sturdier overall design.

C. Sweep Testing

1. Initial Sweep Results

A similar testing schedule was employed for $\Lambda$ testing using the same rapid prototyping technique. As can be seen from the results in Figure 26, the smooth trends are not as predominant in the $\Lambda$ tests as they are for the $\phi$ tests, with uniformity between the three tests showing a very poor trend particularly at any $\Lambda$ back angle beyond about $10^\circ$.

2. Results and comparison to Prandtl’s Lifting Line Theory and laminar testing

Comparison with PLLT was conducted. Both results show that there is an advantage to adding $\Lambda$ to the geometry, albeit very minimal, particularly when compared to the addition of $\phi$. By applying the Kucheman modifications to the $a(\theta_0)$ calculation there was an optimum result observed, with $a(\theta_0)$ peaking at $20^\circ \Lambda$ as seen in Figure 27. However, this result only gave an increase of 1.7% from the $0^\circ \Lambda$ condition. This result did translate when determining applying the PLLT. However, it only produced a 1.2% increase to the lift (0.0024 N) from the $0^\circ \Lambda$ condition.

The relationship between the CFD and PLLT results was not as good as the $\phi$ analysis (Figure 28). This is despite accounting for the data to be offset due to the change in $\phi$ as observed in Figure 22. Furthermore, the relationship in the results was degraded as the tests were conducted at the lower $\phi$ angles, and as discussed in the previous section, the correlation of the CFD and theoretical data had degraded at the lower end of the analysis.
Figure 26. Effect of varying wing sweep on L/D.

Figure 27. Expected change in the 3D lift curve slope with changing wing sweep based on the Kucheman modified equation (Equation 10).

Figure 28. Comparison plots of the CFD data and the calculated PLLT results for the (a) wing lift force and (b) wing drag force when varying wing sweep.

Similar to the trends observed in the \( \varphi \) analysis, comparing the laminar and turbulent CFD tests cases showed that the two tests produced remarkably similar results for the lift forces experienced by the wing, and also for the drag results, provided that the offset due to the turbulence is accounted for. It was observed that the same offset between the laminar and turbulent drag data seen in the \( \varphi \) analysis for the 0° \( \varphi \) wing remained virtually constant at 0.017 N ± 0.000 N across all \( \Lambda \).

However, the lift curve trend did show great affinity, particularly in shape to the CFD results, but was offset with the difference between the CFD and PLLT being -0.080 ± 0.002 N for all \( \Lambda > -10^\circ \). Similarly, for the drag over the same \( \Lambda \) range, the difference between the CFD and the PLLT (induced drag only) was +0.041 ± 0.002 N and +0.240 ± 0.002 N for the turbulent and laminar CFD results respectively.
3. Conclusions

As expected from the literature (Simons, 1994), \( A \) at low \( Re \) provides little advantage. What becomes evident in this analysis is that the magnitude of inherent uncertainty in the mesh when compared to the possible advantages of adding \( A \) to the wing is too similar to provide a definitive result. Furthermore, the addition of \( A \) also pertains an increased difficulty when designing the wing for standardised production. Hence, considering the minimal advantages the addition of \( A \) may produce, it becomes difficult to justify its use with these results.

The main outcome of this analysis was that there was an evident failure of the current mesh design for sweep analysis. The mesh was tested at further backwards \( A \) up to the maximum limit of -55° as shown in Figure 29. This result gave little insight beyond what was observed in earlier testing. There is a slight trend showing indication of increased \( L/D \) with extra backward \( A \), with the results staggering about that general trend.

D. Mesh re-evaluation

1. Skewness and High Aspect Ratio Effects

For the -1°\( \phi \) case, the sweep test cases of -40°\( A \), -45°\( A \) and -50°\( A \) were untested. This was due to either the Mesher failing to produce a mesh, citing that certain cells in the mesh have exceeded the maximum allowable aspect ratio, or when the produced mesh was imported into FLUENT and no result was achieved due to the poor quality of the provided mesh producing cells within the domain with extremely high skewness.

The cell aspect ratio is a measure of the stretching of the cell and is defined as the ratio of the maximum distance between the cell centroid and the face centroids to the minimum distance between the nodes of the cell. Cell skewness is the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume (FLUENT). Therefore, a large aspect ratio in a tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. Hence, the two separate errors discussed above stemmed from the same problem in the mesh design, where certain regions of the field and the specified cell size constraints conflict, causing the cell to be several orders larger in one axis relative to the others.

Upon analysing the test environment it was deduced that the region of error was most likely to be the vicinity of the trailing edge apex of the wing at the wing root, as with increased \( A \) between the two wing sides decreases. Hence, a conflict between the finer mesh sizings in the proximity of the wing which was refined to provide greater detail in that region and the cruder sizings for the domain away from the wing. It must be noted, there was a mesh created for the -55°\( A \) case. Hence, this suggests that the apex angle was small enough for the finer mesh to prevail in that region. This has severe consequences on the reliability of the mesh results, as it in turn causes difficulty for the results to converge, as the cell aspect ratio is an indication of the relative amount of iterations required for the result to converge to a similar magnitude in the separate directions.

2. Poor convergence

As can be seen from the residual plots in Figure 30, convergence starts to degrade with increased backward \( A \). The standard residual plot taken from the 0°\( A \) results (Figure 30 (a)) is remarkably consistent for the results achieved for all the sweep between 30°\( A \) to about -20°\( A \) and remained fairly consistent for all \( \phi \) tests. These plots are characterised by a distinct drop in all the residuals with a minimum reduction in the scaled residuals of 5 orders of magnitude \([O(10^5)]\) for the turbulence model and 8 orders for the momentum model (velocity residuals). However, slight oscillations start to appear beyond this point as captured in the residual plot (Figure 30 (b)). These oscillations continue to increase, also coinciding with the degradation of the level of convergence which drops several orders of magnitude when compared to the standard A plot. Further degradation in the reliability of the results appears at about -35°\( A \) (Figure 30 (c)), particularly due to poor convergence and increased oscillations in the results. Of particular interest is the residuals for the turbulence model, which reduced by only \( O(10^1) \) and \( O(10^2) \) for the transport variables \( k \) and \( \varepsilon \) respectively for the -35°\( A \) case. These results were, however, redundant considering that the oscillations themselves were about of the order of \( O(10^5) \).
These discrepancies appear to subside with increased $\Lambda$ (Figure 30 (d)) with the results at $-55^\circ\Lambda$ showing a return to similar results to those shown in plot B, albeit with a more pronounced oscillation of $O(10^1)$. Although oscillations generally suggest the possibility of transient effects, particularly considering that there were meshing problems associated with the increased $\Lambda$ up to $-55^\circ\Lambda$ which converged, the problem appeared to be mesh related and not due to the physics setup.

![Figure 30. Comparison of the residual plots for the CFD backward sweep testing at the (a) standard 0° sweep condition, (b) 20° sweep, (c) -35° sweep and (d) -55° sweep.](image)

E. First Refined Mesh (Mesh 2)

A new approach was undertaken in the design of a refined mesh for this analysis, Mesh 2. The main objective of Mesh 2 was to ensure that the trends achieved are not mesh sensitive, and to reduce the anomalies in the results. Although the current mesh (Mesh 1) was already tailored to be fine close to the wing, new sizings were added as outlined in Table 7.

These sizings aimed to address particular problem areas in the mesh design. This ensured that the results would capture any effects in this area, particularly with regards to the wingtips vortices and the boundary layer detail over the wing. However, this resulted in an increase in the mesh size from 500,000 cells to over 5 million cells (Figure 31).

The convergence characteristics of Mesh 2 were virtually similar to that of Mesh 1, exhibiting the same attributes discussed previously, but at extended iteration intervals. The major drop in residuals which determined the time interval had shifted from the 700 iteration mark to about 1,000; and lasted until at least 2,250 iterations. Hence, in order to ensure convergence, 2,500 iterations was set as the determining convergence time. However, this in turn caused the calculation time to balloon to at least over 35 hours, with the higher $\Lambda$ testing taking about 55 hours to converge. Therefore, unlike the previous mesh, the ability to exhaustively test became severely reduced. However, considering the general trends were already set with Mesh 1, only a small subset of the range was required to test the repeatability of the results.

![Table 7. Sizings created for Mesh 2](image)

Figure 32 shows a comparison of the overall $L/D$ results of the old and new meshes. What is evident from the results is that the $L/D$ results for the $\phi$ study are virtually identical. Furthermore, particularly for the lift, the separate component plots appear more refined when compared to the previous plots. The new lift data for example, shows a more linear plot and does not have that slight shift in the results seen in the initial mesh plot for $\phi > 10^\circ$ (Figure 33).
The main advantage of the finer mesh becomes apparent with the \( A \) test as the trend becomes a more defined curve. Even with the reduced data points, the excessive variations observed in the initial mesh for the \( A < -10^\circ \) are smoothed out with the refined mesh. This was at the penalty of an increased mesh size with sweep, with a difference of about one million cells between the 0\(^\circ\)A test case and the -55\(^\circ\)A test case. This confirmed that the convergence in the first mesh with increased \( A \) was due to conflicting mesh sizings.

Furthermore, the finer mesh does confirm that there is a slight advantage for employing sweep as suggested by the initial mesh. However, this increase is much lower than what the previous study suggested it would be, with an increase of only 0.181 \( L/D \) (5.9\%) between the 0\(^\circ\)A condition and the maximum 55\(^\circ\)A, equating to about 0.1\% increase per \(^\circ\)A. This result did again show that there was little advantage in employing \( A \).

Figure 31. Comparison of mesh densities for (a) Mesh 1 and (b) Mesh 2.

Figure 32. Comparison of different CFD L/D results from Mesh 1 and Mesh 2 by varying (a) wing twist and (b) wing sweep.

Figure 33. Comparison of lift results given from the different meshes.
A. New Mesh Design

Despite the successes of the new mesh, upon further analysis there were several key issues with the mesh that needed to be addressed. Through discussions and an in-depth analysis of the results with Dr John Young certain aspects of the results were of concern.

1. Poor wing resolution

Although it was previously recommended that for importation requirements into CATIA, only 20 points from the Excel macro would be sufficient to produce an acceptable spline and loft of the wing, closer inspection of the wing in the ANSYS Mesher, it was shown that the surface of the imported geometry comprised of a combination of jagged edges which were of a comparable size to the mesh size in that area (Figure 34). This would suggest that there was a possibility that the result would be more turbulent than the real case as the air may not necessarily flow smoothly over the surface. This issue was easily resolved by altering the Excel spreadsheet to accommodate 200 data points for each spline. This in itself had very little impact on the mesh size as no mesh sizings were changed.

2. Computational timing

The 40 hour computational time required to produce what was considered an acceptable result that would sufficiently achieve a decent prediction for the \( \Lambda \) did appear to contradict the rapid prototyping methodology, and a new outlook on the mesh was required. Firstly, a symmetry boundary was introduced to the system at the wing root, effectively halving the amount of cells in the analysis as the program was only required to analyse one half wing. This function places a plane of symmetry at the wing root and assumes that the flow is symmetrical about that plane. This concept was briefly entertained in the initial design but was dismissed on the premise that there might be wake effects due to the trailing edge wing apex. However, as can be seen in Figure 35, no such wake existed. This is because the turbulence in the wake is here being modelled via a turbulence model, so is not actually showing up in the simulation as anything other than a local increase in effective viscosity, and hence does not break the symmetry of the flow. Force analysis in the y-direction from the previous mesh studies did however show a slight imbalance in the lateral force of the wing in the order of 0.0002 N (less than 0.1% of the drag force), but this was deemed to be a negligible result. Hence, particularly considering that no asymmetrical wings were to be tested, nor was there any yaw or roll testing required in the evaluation, a symmetry plane became ideal as it was unnecessary to solve the same flow twice as it was mirrored about the wing root.

A second improvement to the mesh, was converting the tetrahedral mesh domain imported from the ANSYS Mesher to a polyhedra mesh in FLUENT. This process decomposes each tetrahedral cell into multiple sub-volumes called ‘duals’ which are associated with one of the original nodes of the cell (Figure 36).

---

**Figure 34.** Close up of the mesh near the wing leading edge showing poor wing design.  
**Figure 35.** Dynamic pressure distribution behind the wing showing the symmetrical wake.

---

**Figure 36.** (a) ‘duals’ denoted as the shaded regions in the blue tetrahedral mesh and (b) the final polyhedral cell.
(a)). Each dual is agglomerated into polyhedral cells around the original nodes. Hence, the duals from all cells sharing one particular node makes up each polyhedral cell (Figure 36 (b)), making the node within the polyhedral cell no longer required and it is removed. (FLUENT, 2006) This process is slightly altered in modelling the inflation layers, where the cells are only decomposed in the plane of the boundary surface, but not in the direction normal to the surface. Hence, the resulting polyhedra will preserve the thickness of the original wedge/prism cells (FLUENT, 2006).

Converting the mesh to polyhedral had a major influence on the resultant mesh with excessive reductions in the overall cell count reducing from the 2.5 million cells in the symmetry setup to about 600,000; a total cell reduction of about 76.5% (Figure 37). Furthermore this procedure effectively diffuses the high cell skewness and aspect ratio problems by removing the node that skews the cell. However, this process did add one extra step in the setup process and required an addition of 5-10 minutes in computational time initially in FLUENT before starting the testing in order to convert the mesh.

![Figure 37. Comparison of mesh densities for (a) the tetrahedral and (b) the converted polyhedral meshes.](image)

3. Combined improvements of the New Mesh design

The overall combined results of these improvements to the new mesh essentially produced a user friendly mesh design. The new mesh more accurately portrayed the test wing by smoothing out the wing profile design and by keeping the increased mesh density at the vital wing areas that were introduced in the second mesh. This result also provided more reliable results when compared to the initial mesh; this was observed in the increased convergence in the results, with all residuals reducing by at least nine orders of magnitude for the turbulence model variables and $O(10^{-13})$ for the momentum model, with virtually no oscillations observed in the residual plots. Furthermore, the previously poorly converged high $A$ cases converged better than the standard cases.

Although the new mesh was more refined, after converting the domain to polyhedra, the final cell count was only about 20% greater than the initial mesh. This, in turn, meant that the computational time for each iteration was slightly higher than the original mesh. However, as the cell aspect ratio issues were diminished due to the polyhedra, convergence (at the increased accuracy discussed previously) was achieved in only 1,000 iterations and hence the overall computational time required for each test case was virtually comparable to the initial mesh, even with the added mesh conversion time. Furthermore, based on the mesh refinement study analysis, this new mesh reduced the uncertainty to only 0.53%.

However, not all the impacts of the new mesh on the results were positive. The increased cell size had an inherent adverse effect on the analysis time. Other than system results such as the lift and drag data of the wing, all other results which require the use of comparison of separate cell results such as determining the $y^+$ plots and other graphical plots were computationally exhaustive, pushing the computers to their capacity. Hence, the data collection time for the one test had increased from the original mesh from 3-5 minutes to over 15 minutes for the new mesh. It was therefore decided that this was impractical, and hence only the force data and the residuals were collected for these results except for certain exceptional cases which were used for comparative purposes.

B. Validating Mesh Changes

1. Testing procedure

There were 3 separate tests undertaken to verify the results from the previous mesh and to analyse the effect of each of the separate changes on the overall result. These tests were based on the half wing domain based on the Mesh 2 sizings:

Test 1: Use the tetrahedral mesh with a symmetry plane to use for comparison.
Test 2: Convert the domain into a polyhedral mesh but define the symmetry plane as a wall to assess the effect of close walls to the geometry.
Test 3: Use the polyhedral mesh with the symmetry plane to compare with the results in Test 2 to observe the effect of the converted mesh on the results.

The only change that remained constant between these three tests was the use of the refined 200 point wing geometry. The effects of this alteration to the setup were not tested as the old geometry posed meshing difficulties in ANSYS particularly with the interaction between the wing and the symmetry wall. However, the effect of this change was to be extrapolated from the various tests.

2. Testing Results

Figures 38 (a) and (b) depict the results of the testing procedure discussed. The results conclusively confirmed the basic trends previously observed. For all three tests, the difference between the tetrahedral and polyhedral meshes varied minimally. The three curves showed that the correct polyhedral mesh gave the highest value of all three tests for any given wing configuration. The polyhedral mesh that was incorrectly setup with the wall, was consistently the lowest result between the three tests, and had an L/D which was on average 0.043 lower than that of the correctly setup mesh, a variation of only 1.4% in the $A$ results. This difference is due to the +0.0006 N increase in drag caused by the interaction of the wing and the wall at the wing root.

Similarly, the tetrahedral results were slightly lower than the polyhedral results by only 0.014 (0.5%). This therefore confirms that the polyhedral mesh will produce virtually identical results when compared to the tetrahedral results. Hence, considering the advantages that applying this method produces, conversion to polyhedral mesh was considered ideal. The second conclusion from the testing was that although the interaction between the wing and the wall at the root did produce slightly lower results and should be avoided, it had minimal effects on the results.

However, there were two slight discrepancies when comparing the old tests and the new results. The first difference was that for the $\phi$ results the curve at the higher end appears to be flatter than previously observed at higher $\phi$. Up until $9^\circ \phi$, the new results were slightly lower than the initial curve, but unlike the previous results the new mesh plateau slightly and reduce slower after peaking. This difference, although small, does suggest that the setup is slightly less sensitive to adverse pitch.

The second and more obvious difference however, appeared in the $A$ testing where all three new tests deviated from the results from the Mesh 2 study. Unlike the trend found previously, the effect of $A$ on the wing $L/D$ did not rise asymptotically towards a maximum at $-55^\circ A$. Instead, the results show the $L/D$ increasing with higher backward $A$ towards a distinct peak at $-40^\circ A$ before decreasing rapidly. Furthermore, although at $30^\circ A$ forward $A$ all the results are comparable, the new $L/D$ values along the curve increase at a greater rate than the Mesh 2 results, with a maximum variation occurring at the $-40^\circ A$ peak with a difference between the two sets of data of 0.238 (7.4%).

**Figure 38. New Mesh L/D results for varying (a) wing twist and (b) wing sweep.**

Analysis of the separate lift and drag plots for the $A$ is shown in Figure 39. These plots explain the change in the characteristic curves in the $A$. Of particular interest is the lift plot which undergoes a very similar change in shape as the $L/D$ curve. This new lift curve bears greater resemblance to the PLLT prediction for the $A$ plot and depicts the peak and the sudden drop with greater $\Lambda$. This difference is attributed to the new refined wing design.
as previously the poor wing design caused a reduction in the lift at higher $A$. Furthermore, although the adjusted PLLT greatly resemble the results for the forward $A$ tests, with increased backward $A$ the curves start to differ. Firstly, the rate of increase in lift from the PLLT predictions is not as great as what the results show. Secondly, the PLLT peaks at about -20° while the test results peak at -40° results. The combination of these two differences means that there is a greater change in lift seen in the test results than in the PLLT when comparing the maximum lift to the standard condition.

The drag analysis however depicts a slight discrepancy with a distinct drop of 0.007 N between 0° and -15°, before the curve stabilizes and follows the predicted curve. The reasons for this drop are unclear. However, it is suspected that it is more likely due to an increase in profile drag due to the protruding wing tips into the flow at the 2° angle of attack. This explanation is based on the fact that the deviation does appear to affect all tests with the wing swept forward, while all swept back wings appear to follow the expected trend dictated by the PLLT.

Furthermore, although this drop is not characteristic, and although it is a 16.7% change it does not appear to have an effect on the $L/D$ results due to the great disparity between the lift and drag results. Even if this error is adjusted the difference in the results is less than 1%.

The comparison plots of the New Mesh CFD data and the calculated PLLT results for (a) wing lift and (b) wing drag force when varying wing sweep.

C. New Testing

Several new tests were therefore conducted on this new mesh to analyse more refined aspects of the design and to ensure an optimal result is achieved.

1. Sweep at increased twist angles

A test was conducted to analyse if there is an amplified effect of the increase in efficiency due to applying $A$ at a higher $\phi$. Compared to the predicted PLLT results, the trends are consistent, albeit offset by 0.05 N (Figure 40). Both sets of results show the lift peaks at the same $A$ and with similar increases in the lift. There is, however, a more distinct sudden drop in lift with increased $A$ at higher $\phi$ observed in the CFD results which are not present in the PLLT plots suggesting that since the wing is closer to the stall condition at the higher $\phi$, the extra $A$ promotes the stall further. Furthermore, although at the higher $\phi$ condition there is a 3.8% increase in lift by sweeping back to -40$^\circ$A when compared to the 0$^\circ$A condition, the analysis of the $L/D$ results shows that this only translates to an increase of 1.3%.

2. Twist at increased sweep angles

A similar test was conducted to analyse the effects of varying $A$ on the $\phi$ variation plots (Figure 41). The results show that by employing the ideal -40$^\circ$A, there is an initial definite improvement in the $L/D$ (8.4% at 0$^\circ$A) that diminishes with increased twist up to the optimum 11$^\circ$A condition where the results are identical. For greater values of $\phi$, however, the $L/D$ for the optimum $A$ is less than the 0$^\circ$A condition. The reason for this effect is wing stall due to separation in the outer segments of the wing reducing the wing lift in this condition.

This result suggests that for low $\phi$ the addition of $A$ is highly favourable; however, the combination of high $\phi$ and high $A$ can be detrimental to the results on the wing efficiency.
3. Twist at varied angles of attack

A test of the $\varphi$ at varied $\alpha$ was conducted to view the variations to the curve. As expected, the lift did distinctly produce an incremental increase with each degree of increased angle of attack. Furthermore, the maximum $L/D$ did shift to lower $\varphi$ with increased wing $\alpha$ (Figure 42). The results indicate that curve does shift across. However, there was a slight increase in the maximum $L/D$ observed with increased $\alpha$ and peak flattens and becomes less distinct. Therefore, this meant that the wing at higher $\alpha$ produced similar $L/D$ over a wider range of twist angles. Hence, it is advantageous to choose a $\varphi$ that is slightly lower than the optimum as this will increase the stability of the wing when the $\alpha$ is varied by a couple of degrees as it becomes less sensitive to the pitch. In this instance, a twist angle of 9$^\circ$ for the wing is ideal as it can travel in the $\alpha = 2^\circ$-$6^\circ$ region with a maximum change of $L/D$ of only 2.7%.

Figure 40. Comparison of lift results at different twist angles using (a) PLLT and (b) CFD.

Figure 41. Comparison of the L/D results at 0$^\circ$ sweep (standard) and 40$^\circ$ sweep (optimal)

Figure 42. Effect of changing the angle of attack on the twist results.
4. Optimum twist line

Being the most influential variable in altering the $L/D$ of the wing, it was important to determine the optimum point along the chord to impart the twist. Although theory suggests that the most appropriate position is about the quarter chord point, initial testing using the cruder mesh did suggest a slight bias towards twisting about the trailing edge over the leading edge but this was inconclusive. However, by using the new mesh, this hypothesis was retested. Tests of the wing at various $\phi$ were conducted with the position of the twist line tested at the c/4, leading edge and trailing edge. Based on the assumption that the c/4 would be the optimum result, the difference between the c/4 results and results at the other two twist line positions was used as an indication.

The results in Figure 43 show that particularly for higher $\phi$, having the twist line at the trailing edge was the most efficient, producing an increase of 1-2% over the c/4 result. This was an ideal result as by twisting from the trailing edge the maximum dimensional height in the Z direction of the wing is reduced, albeit being by a maximum of a couple of millimetres.

5. Viscosity Ratio Limiting Tests

As previously discussed in the turbulence model setup, although FLUENT calculates the values of I and $\beta$ at all the cells, it is important to have an indication of what these values are. Particularly considering that the suggested spectrum of $\beta$ was rather large ($10 > \beta > 1$), tests were setup, purposely limiting the value of $\beta$ in the inlet and outflow boundary conditions from the initial 10 to 6 and 1. The only apparent result of this experiment was that FLUENT warned that the viscosity ratio was reached on average 2 and 16 cells for the $\beta = 6$ and $\beta = 1$ tests respectively. Similarly, comparison of the data showed that the lift and drag results varied by 0.1% or less for each of the cases which were expected considering the few cells that were affected.

It was therefore determined that the result was insensitive to this parameter, however, in order to negate having this warning reappear in later tests.

D. Finest Mesh

Although the results appeared to be conclusive with the new mesh, there was one main anomaly in the results which was a cause for concern. Considering that for low speed flight pressure is the most dominant contributor to the lift and drag forces (Katz, 2001), one would expect that the pressure distribution on the wing to be very uniform. There were unexpected variations in the distribution, with lines of equal pressure not running straight across the wing as expected but rather traced out unexplainable paths as shown in Figure 44.

This result was of concern and, although it most likely due to the high cell growth rates and considering the results of the mesh refinement study should bear no resemblance on the lift and drag results, a highly refined mesh was setup that halved all the growth rates, decreased the wing surface sizing by a factor of ten and increased the number of inflation layer to 20 (Table 8).

This new refined mesh was very problematic as it pushed the processing limits of the computers used. Starting with a tetrahedral mesh of over 20 million cells these meshes required over 8 hours of computational time to convert to the polyhedral mesh which had a size of the order of 5 million cells. The computational time required to reach convergence was of the order of 40-50 hours however the reduction in residuals was only $O(10^5)$. In all 11 tests with this mesh at various wing geometries were conducted, producing results that virtually identical to the results achieved in the New Mesh, with variations of less than 4% noted. Unfortunately, confirmation that the pressure anomalies have been solved could not be acquired due to FLUENT continually crashing when requested to generate the graphic.

![Figure 43. Optimum twist line analysis plot](image)

**Table 8. Sizings created for Mesh 2**

<table>
<thead>
<tr>
<th>Sizing Type</th>
<th>Selection</th>
<th>Size</th>
<th>Growth Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Face Sizing</td>
<td>Wing Surface</td>
<td>2.5e-4 m</td>
<td>1.05</td>
</tr>
<tr>
<td>Edge Sizing</td>
<td>Leading Edge</td>
<td>2.5e-3 m</td>
<td>1.05</td>
</tr>
<tr>
<td>Edge Sizing</td>
<td>Trailing Edge</td>
<td>2.5e-3 m</td>
<td>1.05</td>
</tr>
<tr>
<td>Edge Sizing</td>
<td>Wing Tip Edges</td>
<td>2.5e-3 m</td>
<td>1.05</td>
</tr>
</tbody>
</table>

![Figure 44. Poor surface pressure plots](image)
VIII. Collaboration

The final step in the design process was to collaborate the optimized results which were achieved between the parameters discussed here and the work done by James Walduck. Early indications in his testing regime suggested that the lower the wing thickness, the more efficient the wing. Similarly, for TR, a value around 1.0 was the most ideal. As stated previously, for the two variables optimized in this report, Λ had very little effect on the efficiency results overall and in particular with increased φ.

Hence, based on these results two tests were conducted to observe the effect of optimizing the t/c ratio and TR with the results shown in Figure 45. Noting the influential nature of the change in φ on the L/D results, φ was used as the test variable.

![Optimisation Results](image_url)

**Figure 45. Results of collaboration testing.**

The plot shows that the most efficient wings are in the vicinity of φ between 11°φ for the thicker airfoil and 8°φ for the thinner one. These results correspond to the maximum φ for those wing geometries to achieve the maximum effective incidence angle (α + φ) to the oncoming flow at the wing tips before the stall condition as dictated by the 2D airfoil data plots. Despite the increased lift force with increased φ due to the higher incidence angles along the span, stall will start at the wingtips. Hence, employing φ in the vicinity of these optimum angles is ideal, however, not beyond them.

It is evident that particularly for lower φ, the use of the thinner airfoil with the TR = 1.0 wing is the most ideal result. However, the lower the t/c ratio produces a more pronounced crest at a slightly lower maximum L/D when compared to the other airfoil geometry. This is due to the thinner wings being more susceptible to flow separation at the segments which experience higher φ and α. Furthermore, there are inherent manufacturing difficulties and structural rigidity concerns when using the thinner airfoil. Hence, although the thicker wing geometries are less efficient, their ability to keep the flow attached makes them more ideal. Furthermore, the thicker wing is more structurally sound and has the potential to have the power cells and electronic circuitry stored within it similar to the Wasp design, removing potential sources of increased drag which is not accounted for in this study.

The effect of employing a wing with TR = 1.0 on the results is remarkably similar to the effect of Λ on the thicker airfoil, providing an increase on the L/D results which diminishes with increased φ, and is negligible in the twist region of interest. However by changing the TR, the lift distribution is altered and is less likely to mimic that of the ideal elliptical wing. Employing the lower TR does nonetheless, increase the difficulty of manufacturing the wing, as the rectangular wing is simplest. Another advantage of the AR = 0.5, is that at the higher φ used, when α is increased and the wing starts to stall at the wingtips, the smaller cords in these regions will ensure most of the wing area will still be producing lift. It must be noted that outboard stall will rarely be symmetrical and hence there is a tendency for it to induce rapid roll unless these control issues are addressed (Simons, 1994).

Therefore, in conclusion, it was found that the most ideal wing would be the thicker NACA 0008 wing at 0°Λ with TR = 0.5 and 9°φ which is depicted in Figure 46.
IX. Wing Manufacturing and Wind Tunnel Testing

Initially under the proposed project plan, the validation analysis was expected to be done through the use of experimental data. The method involved the use of the ADFA low speed wind tunnel to undertake experiments on models which were built using the balsa CNC lathe to evaluate the drag and lift forces, and compare this with the CFD results.

There were however several problems encountered with regards to this proposed methodology. The predominate barrier however was that there was insufficient time to attempt this testing phase due to several problematic junctions reached with the mesh which blew out the testing time required. It was deemed that the correct mesh setup was core to the project as it would provide a setup which could be used as a framework for further reference.

With regards to the manufacturing, the setup was simple as the whole process was used CATIA, and hence the wing geometry was simple to input using the same process used. An initial wing was setup as shown in Figure 47. As can be seen, there are extra portions of the balsa piece (grey) being machined that need to be designed into the wing (yellow) to ensure that the wing is still secured to the claps (pink) once the machining is complete. Furthermore, there are inherent tolerance limitations of the machining process due to the machine being used and the balsa wood properties. It was understood that the finish on the balsa would not be fine and will require sanding and lacquering to produce the smooth wing surface required for testing. Also, although the softness of the balsa wood is advantageous in that the milling process is quick, the balsa wood cannot be milled to a thickness of less than 3 mm as this would cause the wood to splinter. Hence, for the aerofoils required it appeared that there is a requirement for finer manufacturing tolerances than what could be provided using the balsa CNC lathe, particularly to the small thickness of the wing.

Experimentally, there were also problems faced with designing the stinger required to hold the wing in place in the wing tunnel. The stinger is required to provide both support in all three axes of the wing to ensure that there is no bending moment imparted on the stinger due to lack of stiffness, whilst also being non intrusive so as to not affect the flow around the wing. This however was challenging due to the limited cross sectional area of the wing, which, unlike the model F1 car designs tested by Kell (2009), do not have a large cavity in the back for the stinger.
X. Conclusions

This report details the work undertaken to design the wing for the new school MAV using CFD as a rapid prototyping technique. The 150 wing geometries used in over 600 tests requiring almost 3,500 computational hours alone is merely an indication of the exhaustive approach undertaken in this analysis in order to come up with the detailed results.

The work was built up around establishing a systematic and repeatable testing framework based on a standard setup in FLUENT that enabled comparative results to be produced, and analysed the effects of altering the geometrical parameters of the wing. Four main geometric parameters were used in the optimization study, namely, the AR, λ, φ and Λ with the report concentrating predominately on the latter two. The wing design required a parametric model to be produced and that enabled the variation of the geometric parameters. However, due to the inability of ANSYS or CATIA to provide such functionality, an Excel program was setup to manipulate a set of data points that describe the NACA0008 aerofoil used geometric correlations to produce the airfoil sections at three wing half span stations. The resultant airfoil sections were then imported into CATIA using a macro that also splined and lofted the results into a wing surface which was then closed and then imported into the CFD domain.

An initial mesh was derived and tested for a test case at 0°φ and 0°A, giving an uncertainty level in the L/D of just over one percent from the extrapolated results. The PLLT was used as verification tool, using the modified lift curve slope equations set up by Kecheman and Helmbold to account for Λ and the low AR respectively. However, although providing credible trends and results comparable to the PLLT suggested results, after the first round of testing it became evident that the mesh broke down with increased Λ, as the results started to veer from the established trends and the results showed poor residual convergence. Hence, a new finer mesh, Mesh 2, was set up with sizings placed in specific areas of interest such as the wing edges and surface. The new mesh confirmed the predicted trends of the previous mesh and most of the previous results bar the backward sweep data.

The extra sizings introduced into Mesh 2 did, however, increase the mesh cell count and consequently the convergence time by over tenfold, making it impractical for the rapid prototyping requirements. Hence, a new mesh based on Mesh 2 was created using a symmetry plane and a polyhedral mesh which reduced the convergence time considerably. Also a refined airfoil shape was also introduced. This resulted in the most reliable results with distinct trends in all tests conducted. The New Mesh appeared to show accurate results as well, following all the predicted trends (although offset slightly) from the PLLT results, in particular, the distinct peak in the lift due to sweep.

Finally, a collaboration study was conducted to combine the trend results obtained by James Walduck for the wing TR and λ to the Λ and φ trends to produce an airfoil that is optimized in all four parameters. Practicality and ease of manufacture were at the forefront of the decisions under which the aerofoil optimized was deduced. Hence, the thicker NACA0008 aerofoil design was chosen over the NACA0004 due to structural integrity and separation issues at the higher α.

However, as the maximum advantage of any of the four parameters on the aerodynamic efficiency of the wing was by varying φ, optimum φ was key to the design process, despite its manufacturing difficulty. A wing with 9°φ was chosen as the most optimal due to its pitch insensitivity, despite the maximum L/D being achieved at 11°φ. Backward Λ was dismissed as being viable in this configuration as it provided a negligible increase. Similarly increasing TR also showed a negligible increase in L/D and hence it was left unaltered at 0.5 as it was closer to the elliptical wing distribution.

Although, these results were not physically tested in the wind tunnel, it is believed that the study did achieve the goals it aimed to achieve. The results were completely verified through the mesh analysis and the refinement studies, and partially validated with the PLLT results, which is based on experimental data. It was deemed more important at the time to produce a credible mesh then to pursue experimental analysis due to the inherent room for error. Furthermore, the project was just as much about setting up the framework for continued work in the future design of the school MAV, as it was to optimize the geometrical properties of the wing.

XI. Recommendations

The work being presented in this report was merely a stepping stone to initiate the design of a school MAV. It is hoped that the design work does continue in earnest in the years to come. There are several recommended improvements to help complete this work. Firstly, despite the high confidence in the results due to the exhaustive mesh analysis undertaken, it would be ideal if a model is built and tested to validate the data. Once this is completed, it is recommended that another review of the turbulence model setup in FLUENT should take place in order to adjust the results and possibly compare the results with other turbulence models to determine the most accurate one for further analysis.

Although optimized for the four geometric parameters discussed above, this design is not necessarily the most ideal for the MAV. Furthermore, possible stability issues due to the effect of implementing the
recommended high φ and the low TR need to be analysed further, particularly with respect to the pitch stability and roll stability at stall.

Other parameters such as camber, stiffness and roughness will all affect the result of the design. From Figure 48, it appears that our airfoil lies in the Re region where it might be potentially better to employ a thin, highly cambered aerofoil instead. Furthermore, other wing shapes such as the regular or the inverse Zimmerman shape need to be explored.

These changes should be simpler to implement into the design as the methodology and groundwork have already been laid by this project. Further continuous enhancement of the design should be implemented to improve on the results achieved.

Furthermore, work is required for the integration of other components of the MAV into the design, in particular, the propeller in order to determine the effect of the propwash on the wing.

![Figure 48. Effect of aerofoil shape on $c_L$ and $c_D$ at low Re (Chklovski, 2010)](image)

Acknowledgements

First and foremost, I would like to thank God for His grace and granting me the sanity and patience to persevere through this project and the many long nights testing in labs.

Many thanks go to Dr Rikard Heslehurst for your support and supervision despite the turbulent nature of the project. Many thanks for always being available no matter where you were around the globe and in particular for the weekly meetings which, despite being easygoing, provided the direction for the week ahead and always digressed towards admiring your many aircraft models.

To Dr John Young, despite not being your students your advice and guidance very much appreciated. Even though sometimes you were impossible to find, the project would not have gotten off the ground without your assistance. I meant each and every thank you, after each of the sessions in the lab as I watched my mesh being torn apart as the newer mesh was better for it. I can only aspire to be as knowledgeable in the field.

Dr Warren Smith, for his time and effort in helping setting up the manufacturing of the wing in the balsa CNC lathe despite its inevitable demise.

Mr Alan Fien, for his help and support in navigating the quirks of CATIA, and providing the macro without which the project would have analysed a flat plate.

To the SEIT, many thanks for providing me concurrent access to the 24 computers Blg 20 Lab, you'd be pleased to know that they have been put to good use over most nights in the past three months with over 3000 hours of computational time to produce the results presented in this thesis.

To OFFCDTS James Walduck and Ryan Turner whose help and entertainment in the field of CFD over the year have been most memorable. Further thanks to my neighbour OFFCDT Aidan Watters who although was not a member of the CFD clan was always around to provide a laugh when it was most needed.

To Miss Matha Ibrahim and Miss Sarah Mansour, for the many nights of laughter and entertainment and helping decipher meaning from my writings in order to distil it into my report.

Finally, I would like to thank my family and friends back home for their continuous support over not only the course of this project but more made the last 4 years fulfilling and worthwhile.
References


Thesis Report 2010, SEIT, UNSW@ADFA