EXPERIMENTAL ANALYSIS AND ENHANCEMENT OF AIRFOIL PERFORMANCE IN REGARDS TO LIFT-TO-DRAG RATIO AND STABILITY IN A SUBSONIC FLOW BY UTILIZING CFD AND A LOW SUBSONIC WIND TUNNEL

NAPATH HANKAMPA
KANNAWAT VAYUWATTANASIRI

A THESIS REPORT SUBMITTED IN PARTIAL FULFILLMENT
OF THE REQUIREMENTS FOR THE DEGREE OF
BACHELOR OF MECHANICAL ENGINEERING
SCHOOL OF INTERNATIONAL DISCIPLINARY ENGINEERING
PROGRAMS, SCHOOL OF ENGINEERING
KING MONGKUT'S INSTITUTE OF TECHNOLOGY LADKRABANG
2566

THESIS PROJECT OF YEAR 2566

MECHANICAL ENGINEERING, SCHOOL OF INTERNATIONAL

DISCIPLINARY ENGINEERING PROGRAMS, SCHOOL OF ENGINEERING

KING MONGKUT'S INSTITUTE OF TECHNOLOGY LADKRABANG

Project Title Experimental Analysis and Enhancement of Airfoil Performance in Regards to Lift-to-Drag Ratio and Stability in a Subsonic Flow by Utilizing CFD and a Low Subsonic Wind Tunnel

Student Name

1.	Napath Hankampa	Student ID 63011190
2.	Kannawat Vayuwattanasiri	Student ID 63011167

____________________________ Advisor

(Dr. Bumroong Puangkird)

Experimental Analysis and Enhancement of Airfoil Performance in Regards to Lift-to-Drag Ratio and Stability in a Subsonic Flow by Utilizing CFD and a Low Subsonic Wind Tunnel

Napath Hankampa Student
Kannawat Vayuwattansiri Student
Dr. Bumroong Puangkird Advisor
Academic year 2566

ABSTRACT

This thesis presents an experimental analysis of airfoil performance in a subsonic flow environment with a primary focus on optimizing the lift-to-drag ratio and stability characteristics. The study employs both CFD and a low subsonic wind tunnel to conduct a comprehensive investigation into the various factors, such as airfoil shapes, sizes, surface treatments, etc. We intend to study existing airfoil models, examine their capabilities, and identify their flaws in order to design our own model with improved aerodynamic efficiency, which will have direct implications for aircraft design, energy-efficient transportation, and many other aerospace and aeronautics engineering applications.

ACKNOWLEDGEMENT

This thesis was constructed successfully, thanks to the invaluable guidance from our experienced advisor, Dr. Bumroong Puangkird, who helped us through with the project. We would also like to extend our thanks to our parents, friends, brothers and sisters for consistently motivating and supporting us. This thesis would not have been possible without the collective efforts of all those mentioned above. Thank you for being part of this significant chapter in our academic journey.

Napath Hankampa

Kannawat Vayuwattanasiri

TABLE OF CONTENTS

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Statement and Significance of Airfoil Performance in Aerospace and other Engineering Domains

In aerospace and many other engineering domains, the design and performance of airfoils is of utmost importance. One could say it serves as a fundamental cornerstone of aviation technological advancement. Airfoils are specially shaped surface, such as an airplane wing, tail, or propeller blade, that produces lift and drag when moved through the air. The quest for achieving higher levels of performance, stability, and efficiency is a perpetual advancement which forms the evolution of aviation industries. This thesis sought to research and develop an airfoil for operation in subsonic flow conditions, with a specific focus on lift-to-drag ratio and stability. Maximizing lift-to-drag ratio directly influences efficiency, which is a crucial metric in determining the overall performance of an aircraft, thus, making it a focal point in the pursuit of efficient and sustainable aviation. Furthermore, achieving stability is imperative for the safety and reliability of an aircraft, a fact that underscores the significance of our research.

Figure 1.1 Clark Y airfoil of a Curtiss Hawk III

1.2 Research Objective

As previously stated, our objectives are:

1. Investigate the behavior of different airfoil designs under subsonic flow conditions.
2. Identify flaws in existing low subsonic airfoils and discover opportunities for optimizing and enhancing airfoil designs to achieve higher levels of efficiency and stability.
3. Understanding and utilizing both Computational Fluid Dynamics (CFD) simulations and a Low Subsonic Wind Tunnel. Bridging the gap between theoretical predictions and real-world observations.

1.3 Scope and Limitations of the Study

Our research primarily focuses on subsonic flow conditions, so our findings may not be directly applicable to supersonic or hypersonic aviation. Furthermore, the application of both CFD simulations and wind tunnel experiments, while highly informative, has its constraints, particularly in replicating complex real-world conditions. Nonetheless, the combination of these 2 methodologies will offer valuable insights into our airfoil design optimization possibilities.

CHAPTER 2

LITERATURE REVIEW

2.1 Aerodynamics

This chapter will delve into key topics within the field of aerodynamics which will provide the foundational knowledge required to understand airfoil behavior and performance. Aerodynamics is a branch of Fluid Dynamics and is vital to understand how an airfoil interacts with its surrounding air and produces lift and drag forces.

2.1.1 Lift and Drag Forces

In order for an aircraft to rise into the air, a force must be created that equals or exceeds the force of gravity. This force is called lift. In heavier-than-air craft, lift is created by the flow of air over an airfoil. The shape of an airfoil causes air to flow faster on top than on bottom. The fast-flowing air decreases the surrounding air pressure. Because the air pressure is greater below the airfoil than above, a resulting lift force is created. Consequently, the force that opposes the motion of an object through a fluid (air) is the drag force. It is caused by the deviation or difference in velocity between the fluid and the object. In aviation, reducing drag is essential for achieving greater efficiency and speed. The lift-to-drag ratio is used to measure the efficiency of an aircraft or any aerodynamic object. It is calculated by dividing the lift force by the drag force. A higher L/D ratio indicates that an object is more efficient at producing lift and minimizing drag, which is important for aircraft performance. Our airfoil shape design, angle of attack, and flow characteristics all significantly impact lift and drag coefficients, which we will analyze in this research.

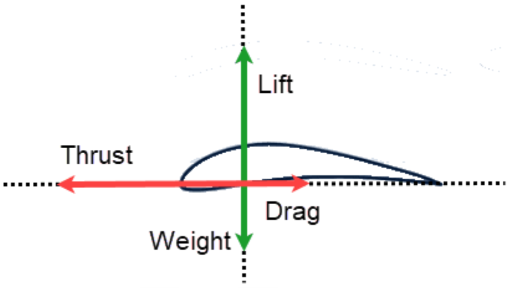


Figure 2.1 Schematics of the reaction forces acting on an airfoil

2.1.2 Mach Number

Mach number is a dimensionless quantity used in aerodynamics and fluid dynamics to represent an object's speed relative to the speed of sound in the surrounding medium, typically air.

The Mach number is defined as:

$$M = V/a$$

where

M represents the Mach number

V is the local flow velocity

a is the speed of sound in the medium

A Mach number of 1 (M=1) indicates that the local flow velocity is traveling at the speed of sound, which corresponds to a sonic condition. A Mach number greater than 1 (M>1) signifies supersonic or hypersonic flow while if an object's Mach number is less than 1 (M<1), it is said to be subsonic. In this range, the object is traveling at speeds slower than the speed of sound. Most conventional aircraft operate in the subsonic regime. We will be working in this speed range.

2.1.3 Reynolds Number

Reynolds number (Re) is another dimensionless parameter critical to the understanding of our research. It is a ratio between inertial and viscous forces and helps predict fluid flow patterns.

The Reynolds number is defined as:

$$Re = \rho u L / \mu$$

where

Re represents the Reynolds number

ρ is the fluid density of the fluid (kg/m^3)

u is the flow velocity (m/s)

L is a characteristic length (m), typically the chord of the airfoil

μ is the dynamic viscosity of the fluid ($Pa.s \ or \ N.s/m^2 \ or \ kg/m.s$)

It is said that if the Reynolds number is low (Re<2000), then the flow is laminar, with smooth, predictable behavior. At higher Reynolds number, the flow transition to turbulence can occur, resulting in increases in both friction and pressure drags over an airfoil surface when separation takes place due to adverse pressure gradient. Numerically, these values are acceptable, however in general, the laminar and turbulent are classified according to range. Laminar flow falls below Re<1100 and turbulent flow falls in a greater range than 2200. Understanding the relationship between Reynolds number and airfoil performance is essential for designing airfoils that are optimized for specific applications and flow conditions. For instance, the airfoil shape and the control of turbulence and separation play a crucial role in managing the effects of Reynolds number on the lift and drag coefficients. However, for this research, we will be working with laminar flow.

2.2 Factors Influencing Airfoil Performance

2.2.1 Airfoil Shape

The design of the airfoil shape is one of the most critical factors influencing its performance that we must consider. Before we begin, it is important to note that there is no single optimal airfoil design. Airfoil design characteristics must be considered when constructing an aircraft, as optimal characteristics can vary depending on the aircraft's intended purpose. For this research, we will be focusing on lift-to-drag ratio and stability in a low subsonic flow condition. There are a few terms that we should familiarize ourselves as the factors that affect an airfoil's performance:

1. Chord line: a theoretical straight line between the leading (front-most edge) and trailing (rear-most edge) edges of the airfoil.
2. Camber: The centerline, halfway between the upper and lower surfaces of the airfoil. The camber describes how curved an airfoil is.
3. Thickness: the thickness effect of the airfoil establishes different characteristics depending on Reynolds number condition. At lower Reynolds number (Laminar flow), thinner airfoils are more optimal in reaching higher maximum lift values. In contrast, at higher Reynolds number (Turbulent flow), the thickest airfoil reaches the greater maximum lift coefficient.
4. Leading and Trailing Edge Shape: Commonly, most airfoils have a curved leading edge and the trailing edge will typically feature a triangle or cusp shape. Streamlining the airfoil, as in designing the airfoil to be curved and

round on the leading edge and slowly tapering it down to the end trailing edge, reduces the chance of eddies to form, thus reducing drag, consequently improving our lift-to-drag ratio and thus, our airfoil performance and efficiency.

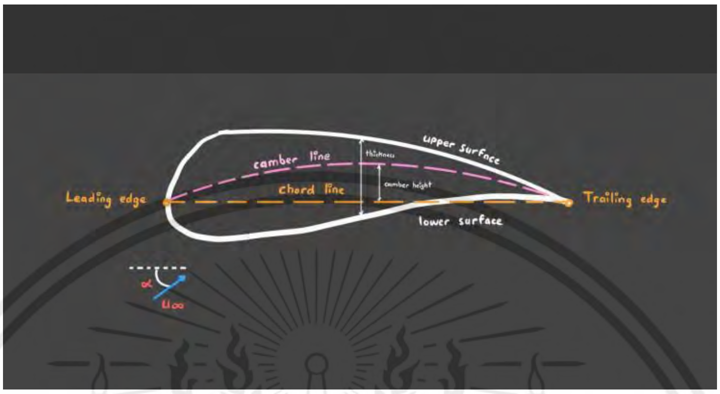


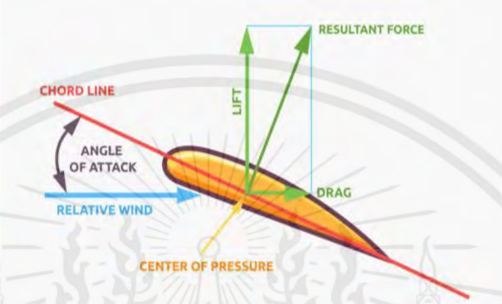
Figure 2.2 Airfoil Basics

All of these factors, more or less, are fundamental to our research and must be considered in our airfoil design. Now, we will examine the environmental factors outside our airfoil.

2.2.2 Angle of Attack

The angle of attack (AOA) is the angle between the chord line of the airfoil and the oncoming airflow. As the AOA increases, so does lift, up to the wing's stall point. However, induced drag also increases when lift is produced. If the AOA is too high, a stall may occur. A stall is when the flow over the upper surface of the airfoil is unable to turn the corner of the leading edge to follow the upper surface and the flow goes turbulent. The lower surface’s lift is increasing but so is the induced drag.

Figure 2.3 Angle of Attack on Airfoil



2.3 Computational Fluid Dynamics (CFD)

The creation of Computational Fluid Dynamics (CFD) programs is a huge stepping stone for the aviation industry. Now, engineers could simulate a prototype before building the physical model for testing in the wind tunnel. CFDs are critical for designing and optimizing airfoils. They help engineers simulate in order to visualize and analyze how fluid (air) behave in various conditions, allowing for better airfoil design decisions and improvements in efficiency and stability. CFDs also reduce the need for physical prototypes and testing, which can be costly and time-consuming. By simulating in a virtual environment, engineers can iterate and refine designs, saving precious time and resources. This could also mitigate risks, as CFDs can be used to assess and mitigate potential hazards and safety risks. They help engineers understand behavior in extreme conditions and under various failure scenarios. That being said, it is evident that CFDs are a crucial tool in advancing aviation technology, and thus, we will be using Ansys Fluent (Flow) CFD extensively in order to gain insights into airfoil aerodynamics and fluid mechanics, evaluating airfoil performance, and optimize them.

2.4 Wind Tunnel

After testing in a virtual world, the next step would be to build our physical prototype and test it inside a wind tunnel. A wind tunnel is a tool used to study the interaction between the object and the moving airflow. The airfoil is connected and held stationary inside the tunnel, and the air is blown towards it in order to copy the actions of the airfoil when in flight. Each wind tunnel is designed and built for a specific purpose and speed range. Thus, there are several different types of wind tunnels, and also several ways to classify them. They can be designated by the geometry of its tunnel, denoted by the speed regime it operates in, and its working fluid. For this research, the wind tunnel is shown in Figure 2.4. The tunnel geometry is observed to be open on both ends and draws air from the outside environment into the test section. This is classified as an open return wind tunnel, in contrast to the closed return tunnel which is an enclosed system and re-circulates its own air through the test section. The tunnel is able to draw in air and simulate a maximum wind speed of around 40 m/s, which is well within the low subsonic speed regime. A butterfly valve is installed to control the speed of the wind. It is recommended to install the butterfly valve behind the test section at the outlet in order to avoid disrupting the air flow. The butterfly valve would still serve its function when installed at that location, due to the Law of Conservation of Mass. Finally, the working fluid is air, so it is a wind tunnel.

Figure 2.4 Open Return Low Subsonic Wind Tunnel

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Computational Fluid Dynamics (CFD)

As previously mentioned, CFDs are instrumental in our research and pursuit of airfoil design optimization. For this study, we will be using Ansys Fluent (Flow), 2023 R2 Student version.

3.1.1 Mesh Selection

Between the C-mesh, H-mesh, and O-mesh, we deem the C-mesh to be the most ideal and suitable grid for airfoil CFD analysis due to the curvature of the grid matching the leading edge of the airfoil, ensuring better flow separation and better convergence. Thus, selecting the C-grid for our analysis is best, compared to the O-grid which has poor quality elements near the sharp trailing edge, and/or the H-grid which has poor quality elements where it splits at the leading edge. The C grid is able to perfectly conform to the airfoil surface and wake, with elements being perfectly orthogonal as Δx goes to zero. However, a more complicated version of the H or O grid with more blocks could be implemented in order to avoid these issues. But in the end, the C grid is the most ideal for airfoils. A C shape in front of the leading edge, and a rectangle behind it. When united, it should resemble the shape of a soft-point bullet. Advantage of this meshing strategy is that it can be adapted to complex geometries, such as an airfoil. The unstructured mesh is preferred more than structure mesh in this case because we work on the airfoil shape which is complex geometry in geometric shape and we need higher resolution mesh for localized regions to properly calculate those regions precisely. We have to compare our result from the simulation with the original result to compare the errors between it to comprehend and conclude that this simulation can be used and adapted with our project.

3.1.2 Geometry

In this research, we intend to simulate and test 3 existing airfoils and use them as a base for our design. However, for this chapter, the procedure will focus on one of our airfoils: The Clark Y. Thus, all dimensions are respective to that airfoil.

1. Download the airfoil coordinates (get the most number of points possible for accuracy, and close the trailing edge to prevent errors in our simulation)
    a. Edit the airfoil coordinates to be compatible for CFD analysis in Excel
2. Run Ansys, and then open the Fluid Flow (Fluent) analysis system into the project schematics
3. In Ansys DesignModeler, import the coordinate file into the program to plot the curves which will form our airfoil geometry
4. Once the airfoil has been generated, create a fluid domain around the airfoil, as discussed, we are selecting a C-mesh domain over an O or H mesh. Dimensions: 15 chords surrounding the airfoil is found to be efficient and gives accurate results, so 7.5 m x2 (airfoil chord length: 1 m)

5. Then, break up the new surface into 4 quadrants (this will be useful for when we want to mesh the geometry as it will be more refined)
    a. Divide them further, with one line intersecting the trailing edge and the other drawn 0.04 m (distance varies depending on airfoil, between 40 and 80 mm, the 80 mm distance gave us a squished mesh which may change the volume a lot from each cell, and the elements are also skewed) from the leading edge and into the airfoil body, and extended vertically to the upper and lower boundaries of our fluid domain. Now we have a total of 6 surfaces which we can mesh separately (this will help the mesh be more well defined and structured)

3.1.3 Mesh

Create an unstructured mesh (as discussed, selection between structured and unstructured mesh)

1. Insert edge sizings
2. Insert face meshing
3. Generate and examine the quality of the mesh (mesh generation is an art form, impossible to be done perfectly)

3.1.4 Y+

Before simulation, other than mesh quality, we also have to check the Y+ value at the airfoil wall. Y+ value is a non-dimensional value that let us know how much of the mesh is capturing the viscous sublayer.

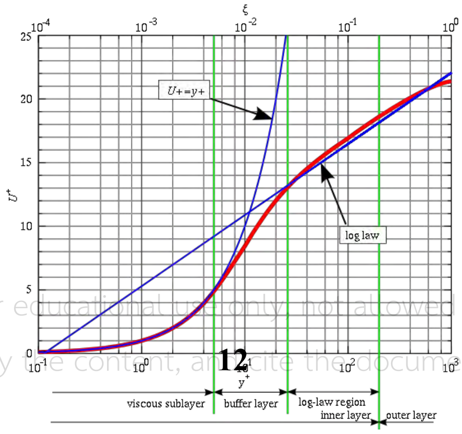


Figure 3.1 Y plus value profile

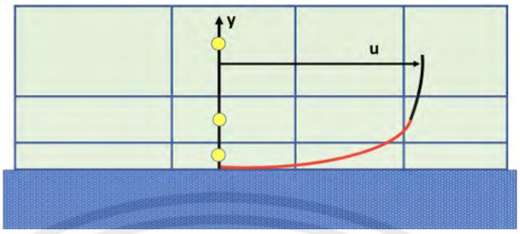


Figure 3.2 Cell height and velocity profile

In order to capture the nonlinear behavior of the velocity profile at the wall, we need to ensure that our first cell height is at a value of Y+1.

Calculations to figure out what our Y+1 value should be, or in other words, how big should the cell at the wall be in order to capture this nonlinear behavior of the velocity at the wall. This is very important for our simulation, if we are concerned about the adverse pressure gradient at the wall, looking at the separation length across the airfoil, etc. (It's important to have a Y+ < 1 for where wall effects are important). However, the Y+ is actually the output of our simulation, because it's a function of the velocity profile (u) which is unknown until the final simulation is solved. Still, we can calculate an estimate of what the Y+ value may be. Before we start our simulation, calculate an estimate of our Y+ value solution, and then acquire the Y+ value as an output to see if we have kept the Y+ value < 1 in the majority of the mesh.

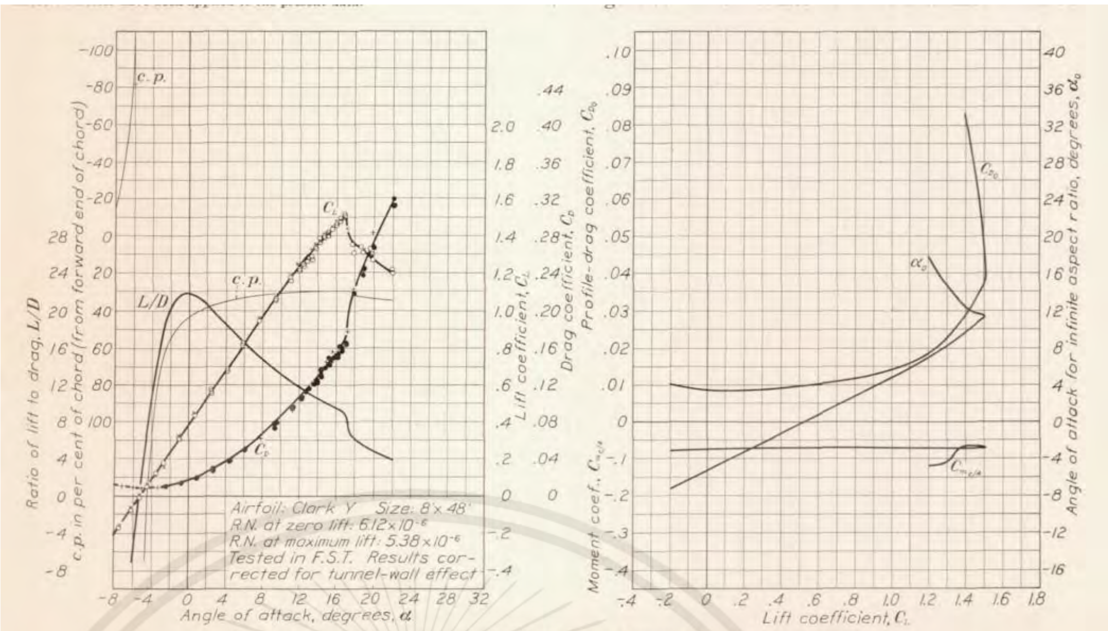
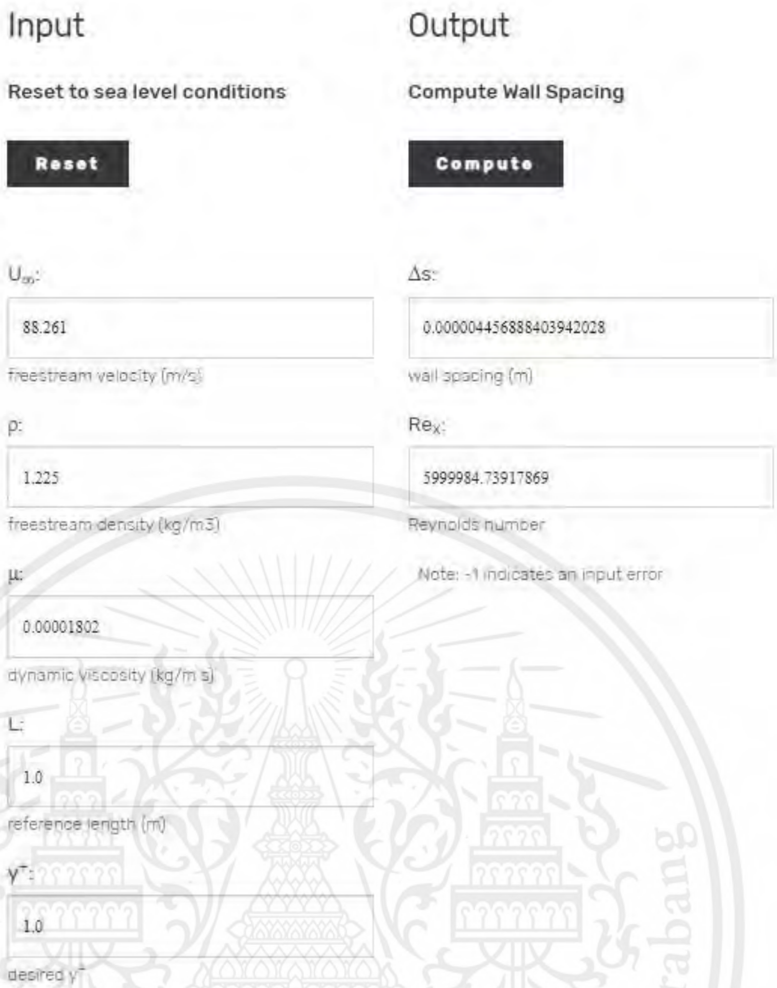


FIGURE 8.—Characteristics of the 8 by 48 Clark Y airfoil at a Reynolds Number of about 6,000,000.

Figure 3.3 Clark Y airfoil experimental data

Source: NATIONAL ADVISORY COMMITTEE FOR AERONAUTICS REPORT No. 502 SCALE EFFECT ON CLARK Y AIRFOIL CHARACTERISTICS FROM N.A.C.A. FULL-SCALE WIND-TUNNEL TESTS BY ABE SILVERSTEIN, 1934

1. Select a Reynolds number for the flow (in low subsonic speed range) compared to an existing experimental data. As we want to work with laminar flow, we selected a Reynolds number of 2100.



U_∞: 88.261
freestream velocity (m/s)

ρ: 1.225
freestream density (kg/m3)

μ: 0.00001802
dynamic viscosity (kg/m s)

L: 1.0
reference length (m)

y^+: 1.0
desired y^+

Δs: 0.000004456888403942028
wall spacing (m)

Re_X: 5999984.73917869
Reynolds number

Note: -1 indicates an input error

Figure 3.4. Volume of our mesh cell, calculated from Cadence Design Systems (CDS)

2. In Cadence Design Systems (CDS), use their Compute Grid Spacing for a Given Y+ for Viscous Flow in CFD feature as shown in Figure 3.4 to calculate for the wall spacing, ΔS (m)
    a. Rearrange the formula for Reynolds equation to find the freestream velocity, u∞ (m/s)
    b. Dynamic viscosity, μ (kg/m.s) of air at $15^{o}\ C$ is 0.00001802 kg/m.s

- The standard temperature in aviation is measured at the mean sea level (msl) pressure of 0.759968 m of mercury (Hg) and is 15° C. Thus, we will use values from that property of air

c. Reference length, L, is the length of the chord of the airfoil, which is 1 m

d. Finally, the freestream density, ρ (kg/m^3) is 1.225 kg/m^3

$$Re = \frac{\rho u L}{\mu}$$

$$u = \frac{Re * \mu}{\rho * L}$$

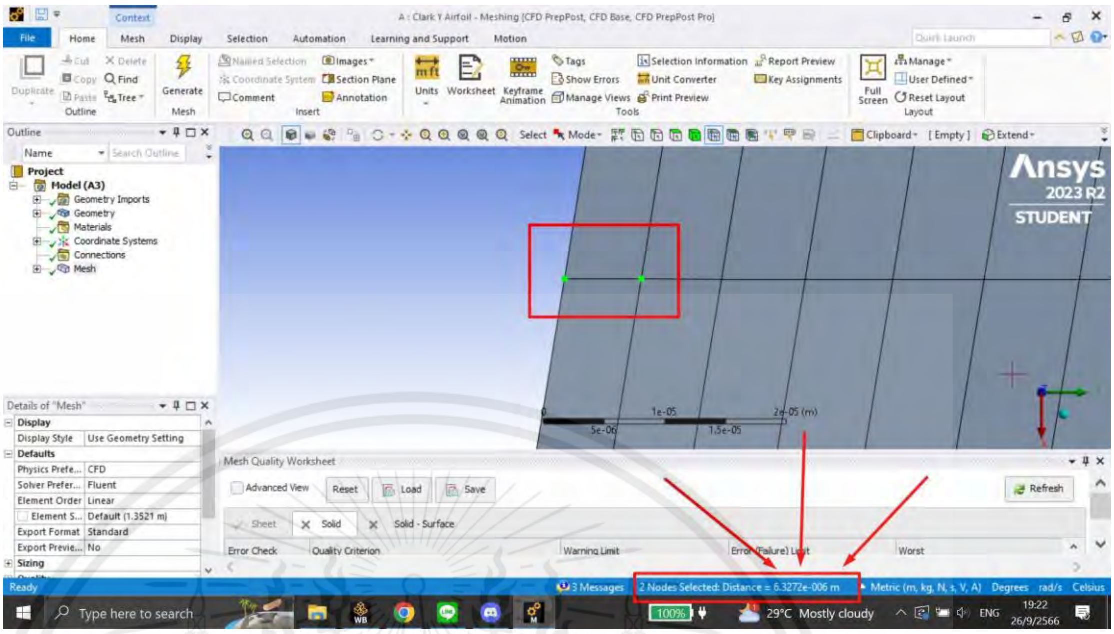
$$u = \frac{(6 * 10^6)(1.802 * 10^{-5} kg/ms)}{(1m)(\frac{1.225kg}{m^3})}$$

$u = 88.261\ m/s$

e. After we have calculated the freestream velocity, u (m/s), check if it's within low subsonic speed range (≤ Mach 0.8 or 273 m/s)

f. Then we insert those values into the CDS program to calculate for the wall spacing, ΔS (m)

g. Wall spacing, ΔS (m) is 0.000004456888403942028 m as shown in Figure 3.4.



Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Prefe...	CFD
Solver Prefer...	Fluent
Element Order	Linear
☐ Element S...	Default (1.3521 m)
Export Format	Standard
Export Previe...	No
Sizing	

2 Nodes Selected: Distance = 6.3272e-006 m

Figure 3.5 First cell height

Select 2 nodes and measure the distance of the first cell height as shown in Figure 3.5. The distance reads $6.3272x10^{-6}$. Compare our wall spacing in Ansys Mesh with the value we got in CDS. Y+ value should < 1.0. This Y+ value is just an estimate, as the Y+ value is actually an output of the simulation. The cell height that we need to maintain a Y+ value of 1.0 will change over the wall. So we must check this after running the simulation to ensure that our Y+ value has remained mainly below 1.0 all around the airfoil wall.

Before exiting Ansys Mesh, we will create our boundary conditions. Our boundaries are named as such:

- The airfoil wall edges are named “Airfoil”
- The front area of the C-mesh domain are named “Inlet”
- The tail edge of the C-mesh domain is named “Outlet”

3.1.5 Simulation Setup

15. Fluent Launcher
    - In options, enable Double Precisions
    - In the Parallel (Local Machine) section, input the number of Solver Processes. The Student Version only allows 4 cores, so if your PC has more cores and logical processors, then you have to purchase their license for the full unrestricted version or crack it. This will affect how fast we solve our simulation. Using a Student version, having 4 or less cores is fine.
    - After we are satisfied with the edited settings, start Fluent

16. In General, input the Solver Type as Pressure-Based, and Time to be Steady

17. In Models, select the Viscous Model as Viscous (SST k-omega) for the airfoil as it's the best model for predicting airflow separation, especially when our Y+ < 1. This model is the most effective in airfoil simulation.

18. Edit our material properties
    - For properties of the fluid, air, check if the density and viscosity are correct. Ansys Fluent will likely give us a viscosity of 1.7894E-5, but as previously explained, we used a different viscosity of 1.802E-5 for aviation simulations

19. In Boundary Conditions, we input the value of our inlet velocity that we had calculated previously. Also make sure that the conditions of our named selections are correct. Inlet is Velocity-Inlet, Outlet is Pressure-Outlet, and Airfoil is a Wall condition.

20. In Reference Values, regulate that the simulation solutions be computed from the inlet. Since we are calculating drag and lift coefficients, we need the values to be the same as the values we used at the inlet, otherwise our drag and lift coefficients will not be calculated correctly

21. For our Solution Method, for Spatial Discretization, set Pressure to either 1st Order or Standard

22. In Residual Monitors, reduce the Absolute Criteria for each to 1e-6 as we are running a very simple simulation so we are fine with restricting more strict convergence conditions on the residuals.

23. In Initializations, we can either use a Standard Initialization or Hybrid Initialization. Solving the finite volume method is an iterative solution, so we need to assign the cells an initial value or an initial estimate, allowing the simulation to proceed and continue. Either initialization methods work fine, they just affect the path to convergence. For this simulation, we used Hybrid Initialization

24. However, before initializing, we have to set up the lift and drag coefficients to be an output of our simulations so we can observe how they are converging.

25. Now that we have everything set, we can run calculations. But first, set the parameters of the calculation. We can set our number of iterations to be a 1000 and that would be plenty enough for the solution to converge, but for this calculation, we are doing 2200. The number of iterations is just to get the best converged solution. Sometimes, we can see that the first few iterations are not smooth, but once it gets to more iterations, the convergence will just stay fixed without change. This means that there is no reason to choose more numbers of iteration than that, as that would only increase calculation time for the simulation without gaining anything from that action

For our Reynold number: 2100 for simulation (Reynolds number from Experimental Data from NASA: 6000000), meaning that our freestream velocity is 0.031 m/s

3.1.6 Adjusting the Angle of Attack

In order to change the AOA in the simulation, we need to enforce components on the flow, for example, change the velocity components such as the X-coordinate, the velocity x cos (α). For the Y velocity of the component, which was originally 0 when we have α=0°, will now be changed to be the velocity magnitude x sin (α). Changing the lift and drag force vectors because they are always measured perpendicular (⊥) and parallel (ǁ) to the relative flow direction. As you can see, the lift is always 90° to the flow direction and the drag is always ∥ to the flow direction. Before, when our α=0°, we defined the lift and drag coefficients but left the velocity and the force components the same. Now they will have to be changed in this simulation as we will change the angle of the flow. For demonstration in this procedure, we will select an arbitrary angle of attack of 8° since that is my lucky number.

27. In Boundary Conditions, go to the inlet velocity and change the X-velocity from the original value (0.031 m/s) to be 0.031 cos (8°) = 0.03069 m/s and the Y-velocity upwards is going to be 0.031 sin (8°) = 0.004314 m/s. If we were to calculate for the magnitude of the X-velocity^2 + Y-velocity^2 = 0.031 m/s

28. We also need to change the force vectors of the drag coefficient to be ∥ and the lift coefficient to be ⊥ with the flow.

- To do that, we go to Report Definitions, then go to our drag coefficient. We want our drag force vector to be parallel with the flow, which is now tilted 8°, so the force vector that's parallel with the flow is

Drag Coefficient Force Vectors: X = cos (8°) = 0.99, Y= sin (8°) = 0.139,

you can see that we can just use simple trigonometry to figure these out

- For the lift coefficient, the X component of the flow is going to be - sin (8°) = -0.139, and the Y component is going to be cos (8°) = 0.99

- After re-initializing our solution, we should get a flow at 8° AOA and our lift and drag coefficients should represent faithfully represent the lift and drag for an 8° AOA

After simulating a different AOA, compare our simulation data with the experimental data. The lift and drag coefficients will be displayed in our console after simulating. We can list our data in excel, calculate for the cL/cD, and then graph a cL vs AOA, cD vs AOA, and cL/cD vs AOA graph as will be seen in the Results and Conclusion chapter of this thesis.

3.1.7 Modifying the Airfoil

Upon analyzing the table of results and graphs from running CFD simulations on both the Clark Y airfoil and the NACA 2412, it is found that the NACA 2412 has better lift coefficient due to its thicker profile. However, this thickness increases the drag force experienced by the NACA 2412. In comparison, the Clark Y airfoil which is more cambered generates lesser drag force, particularly at low AOA, but lower lift generation. The Clark Y airfoil also has a better cruising performance at $\alpha=0° - \pm4°$. To create our own original airfoil, we will use the Clark Y airfoil as the reference base airfoil and modify its properties as such:

- increase the max percentage of the chord by 3% for the same max camber
- increase the thickness by 0.2% for the same chord

This is our initial attempt at modifying an airfoil to fit our performance requirements. There are several ways to implement such changes. One is to write a code in MATLAB and load in the airfoil coordinates to modify them. Regarding which parts of the airfoil to change, Chapter 6: Families of Wing Sections of the book THEORY

OF WING SECTIONS Including a Summary of Airfoil Data By IRA H. ABBOTT DIRECTOR OF AERONAUTICAL AND SPACE RESEARCH NATIONAL AERONAUTICS AND SPACE ADMINISTRATION and ALBERT E. VON DOENHOFF RESEARCH ENGINEER. NASA contains a lot of good information. The OML could then be exported and loaded into either Ansys Fluent. Another simpler method is to import the original airfoil coordinates into XFLR5 (which uses XFOIL for 2D airfoil analysis) or Profili. Simply import the airfoil's original coordinates into one of the aforementioned softwares and edit its properties. Different softwares may accept different types or formats of coordinate files, thus it is important to check the file's properties and contents before importing it into any of the softwares. There are 4 formats for an airfoil coordinate: MSES, ISES, Labeled, and Plain. Most programs accept the labeled or plain format. A labeled format has the string name of the airfoil at the top, sometimes accompanied by the number of points present, followed by lines of the actual content: the coordinates of the airfoil. Only the abscissa and the ordinates are given, so additional z-coordinate points must be added. It is suggested to remove the number for the amount of points from the file, as the program may mistake this number for the first or one of the airfoil's coordinate points. Another detail to take into consideration is the string name of the airfoil. A string name beginning with a number could be mistaken for one of the coordinates. It also cannot be a 'T' or 'F' as this could be misinterpreted as a True or False statement. It is recommended to use Excel in order to edit and format the coordinate. However, it is impossible to use Excel exclusively as some programs demand a space-delimited .txt or .dat file but Excel can only export a tab-delimited .txt file. To fix this, open the tab-delimited text file, open the Find and Replace dialog box (keyboard shortcut: Ctrl + H). Input '^t' on the Find What input and replace them with a single space ‘ ‘. After replacing all, check by choosing an arbitrary coordinate line, place the cursor and use the keyboard arrows to go left and right to make sure that it has just a space in between them.

Once the modified airfoils are generated, they can be imported into any CFD simulations programs or 3D printed for testing in the wind tunnel.

3.2 Low Subsonic Wind Tunnel

The final step is to test our 3D printed prototype design in a low subsonic wind tunnel. We will compare the data and results from this experiment with the ones from CFD and verify them with outside source data.

3.2.1 3D Modeling

To test the airfoils in the wind tunnel, we have to expand into the 3D domain. Any 3D CAD design software can be used to perform such a simple task. In this research, Dassault Systèmes SOLIDWORKS will be used. Before importing the airfoil coordinates into SolidWorks, the dimensions of the wind tunnel test section and the 3D printers must be taken into consideration. The wind tunnel test section length is only 20 cm long, thus the airfoil must be extruded no further than 20 cm. A 2 cm clearance gap is recommended on both sides of the airfoil, leaving 16 cm. Furthermore, the dimensions of the only available 3D printer is only 5.9 in. (150x150x150 mm printing volume). The airfoil has a unit length of 1, and in SolidWorks, the default units were changed from MMGS (millimeter, gram, second) to mks (meter, kilogram, second). This means upon importing, the airfoil will have a chord length of 1 m rather than 1 mm. As mentioned previously, a 1 m airfoil would simply not fit inside a 3D printer or wind tunnel. Thus, the airfoil model must be scaled down. It is determined that scaling the model down by a factor of 7 is the best fit. Merely divide all the coordinate points by 7 to scale it down. Import those scaled down coordinate files into SolidWorks to draw the curves of the airfoil and generate a surface from those edges. The wingroot and wingtip airfoil is the same, as we are only testing one airfoil. Therefore, no other considerations to take, just extrude the 2D surface to expand it into the 3D domain, finally creating our virtual 3D airfoil model. The last feature to add into the 3D airfoil model is a fitting hole.

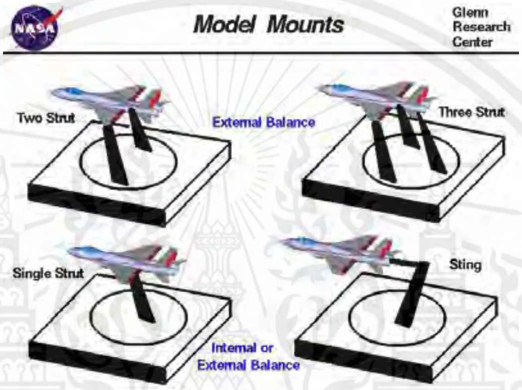


Figure 3.6 Model mounts

The airfoil needs to be mounted inside the test section of the wind tunnel. As shown in Figure 3.6, there are several methods to do so. At the workshop where these tests are conducted, the only available option was the single strut side mounting system. This method is cheaper and poses minimum interference and blockages. They are less rigid than multiple strut mounts and work quite well with internal balance and flow diagnostics. Any reaction forces from the model mount must be measured and considered during data analysis. Interferences from the model mount must be addressed and high blockage ratios can lead to distorted flow patterns and inaccurate measurements. And in regards to structural dynamics, they should be structurally stable and not prone to vibrations or resonance. To do so, the fitting hole for the model mount must be at the aerodynamic center of the airfoil. This can be calculated by dividing the chord length by 4. This is the proper way to do so, however, due to the poor conditions of both the mounts and the fixed supporters, there was no way to hold it in its 0 AOA position when the high wind speeds started blowing. To fix this issue, hot glue could be pasted to fix the airfoil in place. Once the location of the fitting hole is pinpointed, consider how large and deep the hole should be. The model mount at our workshop has a diameter of 6.4 mm (radius of 3.2 mm) and a length of 6.5 cm. However, when printing circles or holes, our 3D printers would always print 0.1 mm less than the specified amount. Therefore, we have to compensate by adding 0.1 mm to the hole diameter (0.64 mm + 0.1 mm = 0.65 mm). Use the extrude cut function to drill a hole into the model. Finally, the virtual 3D models are as shown in Figure 3.7 and Figure 3.8

Figure 3.7 Frontal area of the upper and lower surfaces of the Clark Y airfoil

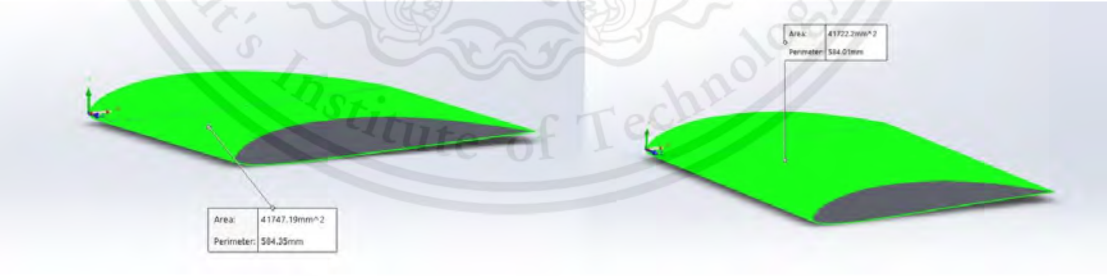
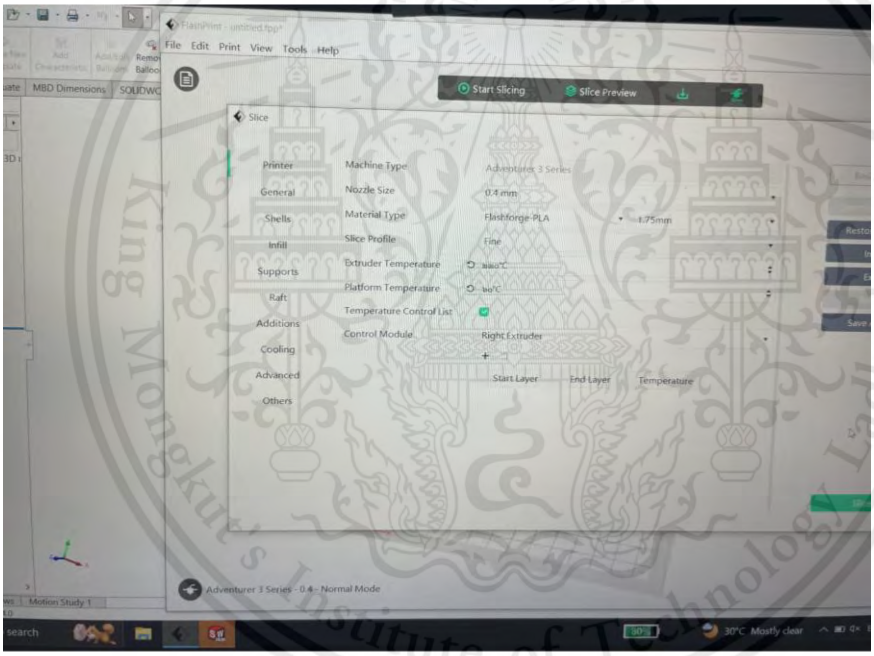


Figure 3.8 From left to right: the thickness and the maximum % chord mods

3.2.2 3D Printing

To prepare a file for the 3D printer, convert (save as) the file from a SOLIDWORKS Part file (.sldprt) to a .STL file format. The slicer software used in this research is FlashPrint 5. Import the .STL file of the 3D airfoil model into that software. To prevent any overhangs and excessive use of supports (which would be tedious to remove and would leave rough surfaces), we will rotate it to have the side profiles of the airfoil laying flat on the surface and printing upwards. Also make sure that the model is oriented such that the fitting hole is at the top. For this research, the details of the machine, settings, and materials are as such:



Machine Type: Adventurer 3 Series
Nozzle Size: 0.4 mm
Material Type: Flashforge-PLA 1.75mm
Slice Profile: Fine
Extruder Temperature: [illegible]
Platform Temperature: [illegible]
Temperature Control List: ☑
Control Module: Right Extruder

Figure 3.9 Slicing settings

Machine type: Adventurer 3 Series
Nozzle size: 0.4 mm
Time: 7 - 16 hours depending on the airfoil model
Material Type: Flashforge-PLA 1.75 mm
Slice profile: Fine
Extruder temperature: 220 °C
Platform temperature: 60 °C

Figure 3.10 Filament

Filament: 1.75 mm PLA + 3D Filament
Color: Fire Engine Red
Print temp: 205-225 °C

Sanding paper was used to remove imperfections such as printing marks and bumps stuck and left over from the printing support in order to prepare smooth airfoil surfaces for wind tunnel testing.

3.2.3 Wind Tunnel

After our physical prototypes are ready, we position them one by one inside the wind tunnel's test section.

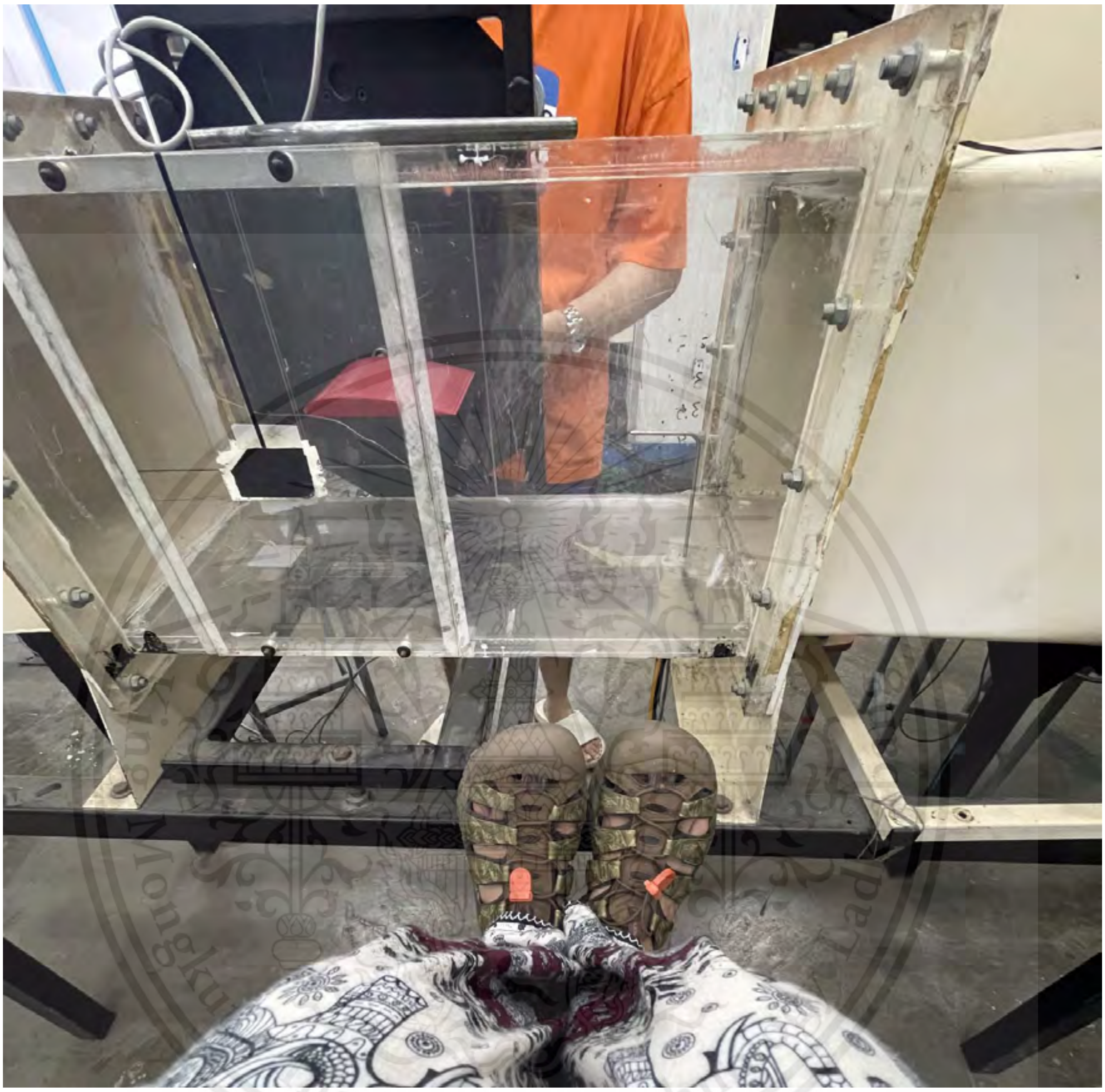


Figure 3.11 Prototype airfoil inside the test section

The black line markings are perpendicular to each other and serve as indicators for positioning the airfoil at a 0° AOA. After the airfoil is mounted and secured, ensure that the test section enclosure's acrylic glass is fastened properly to maintain a controlled environment.

The airfoil's angle of attack can be adjusted manually without disturbing the enclosure as shown in Figure 3.12 below.

Figure 3.12 The manual angle adjustment knob

Since the wind tunnel's butterfly valve was not able to produce a linear increase in wind velocities, the testing procedure then was to just adjust for one specific wind velocity and test all the prototypes in all angles of attack.

Figure 3.13 Butterfly valve

The low subsonic wind tunnel originally came with a multi-tube manometer for measuring the pressure on the models. However, it is difficult to accurately eyeball exactly where the transparent liquid is, inside the manometer's tube.

Figure 3.14 manometer tubes

Thus, we switched to using electronic devices as a superior alternative. We reconnected the pitot tube to redirect its measurements into all of the available electronic meters in the lab.

Figure 3.15 Anemometers

In order to acquire data from the wind tunnel, a program converts ASCII from the load cell into digital numbers to be displayed on the computer. There are multiple softwares available to perform this conversion, but the compatible ones for this research are either Advanced Serial Port Monitor or Docklight. To display these values on a computer or laptop monitor screen, connect the measurement equipment using a USB Type B connector and use the program from Com Port with the transmission speed set to 3600 bps.

The instructions for setting and reading the measurement box is as follow:

HOLD: will hold further outputs and just display the readings at the moment the button was pressed

TARE: calibrates the load cell, setting the starting weight at 0 kg.

KG.(N.): changes the units from kilograms to Newtons. The program originally uses Newtons at the start

AVG: adjusts the load cell results to be the average mean value

Now that we have gathered simulation data from CFD and acquired real-life measurements from the low subsonic wind tunnel, an analysis of said results will lead to this research's final conclusion.

CHAPTER 4

RESULTS AND CONCLUSION

4.1 Results

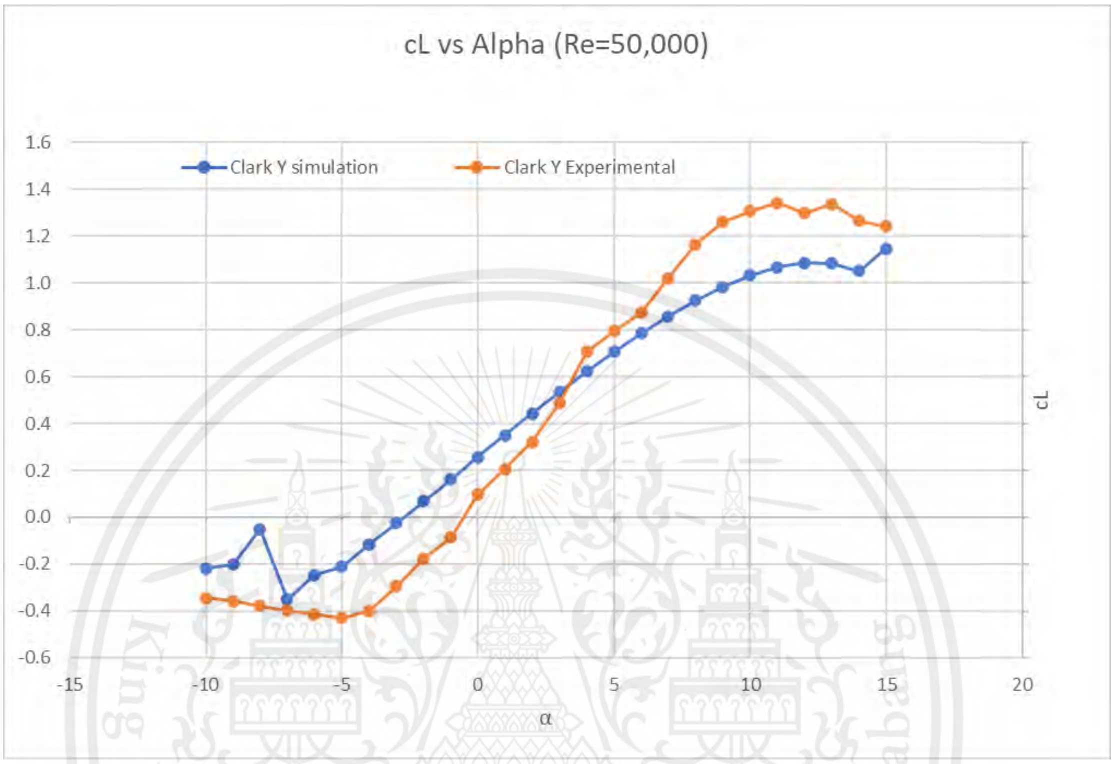


Figure 4.1 Verification of Simulation and Experimental Data

We evaluate our simulation data with experimental data from airfoil tools website which collect the universal data for airfoil experiments to see the difference and accuracy for our simulation data. As a result, it’s acceptable as our simulation setups limits are standard enough for the resource that we have which is satisfied for the research. We can reduce those error gaps by using a setup standard from the organization's procedure that is specialty for this topic.

The point where the largest gap from two data is 79.32%

Airfoil selection and adjustment

We use Clark Y airfoil and our reference model and adjust its component to see how it affects the lift and drag coefficient.

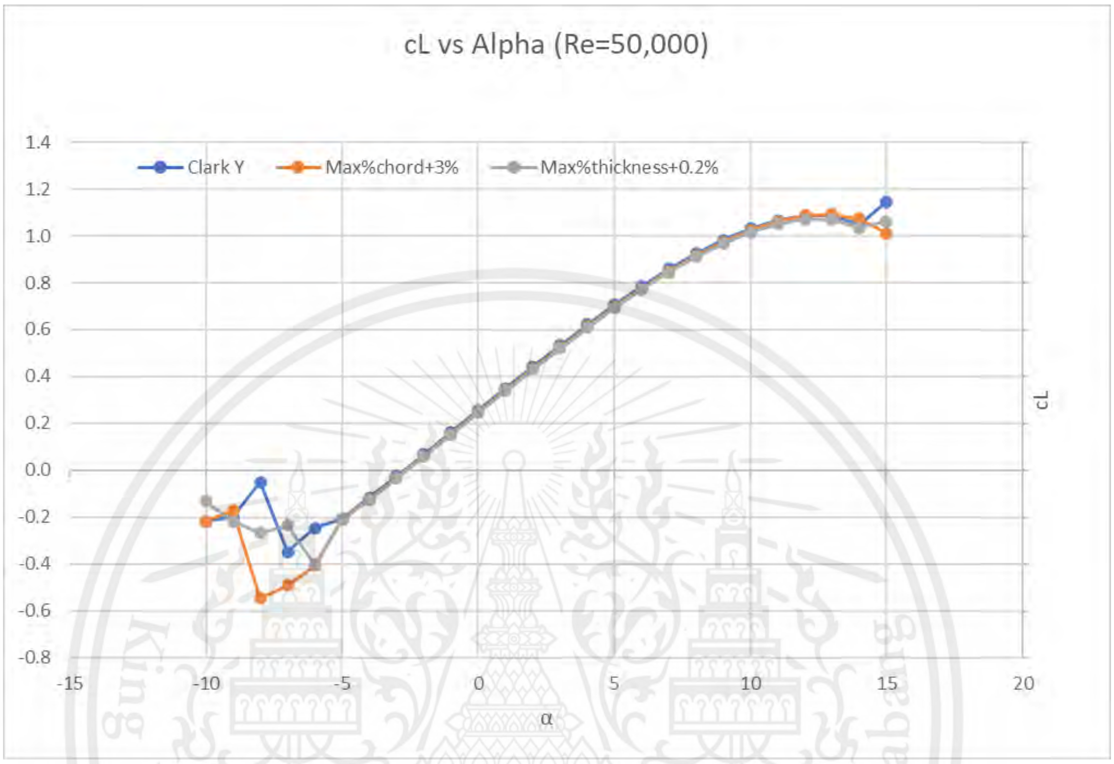


Figure 4.2 cL of Clark Y and its mods

As you can see, The lift coefficients of those models are not significantly visible from the adjustment that we made. The position of Clark Y thickness and percent chord is well calculated and has been tested to give great coefficient lift compared with other two that we included at operation angle. We thought that increasing in thickness may increase in lift coefficient but on the other hand, The separation of flow is better in Clark Y. We also test it with a different velocity in which the condition of the Reynold number is 100,000. The trend of the graph is likely in terms of their performance.

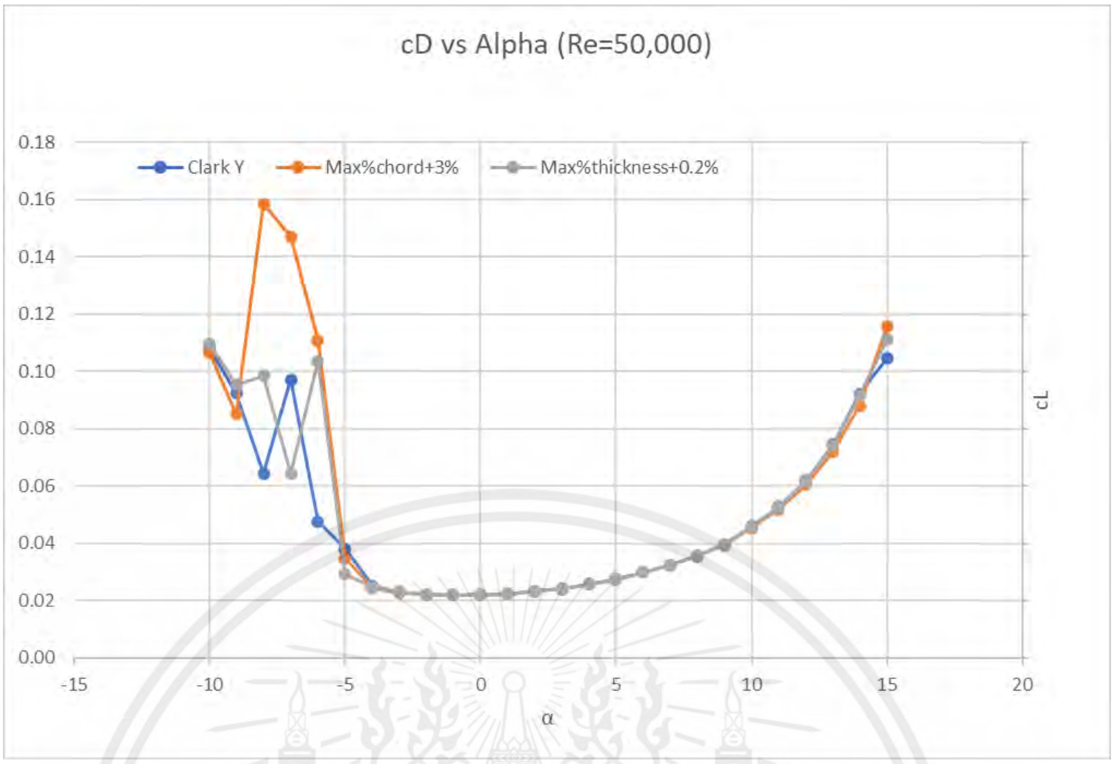


Figure 4.3 cD of Clark Y and its mods

The drag coefficient result shows what we thought it should be as the model that we adjust the chord generate less drag than other models due to the fact that it delay the flow separation so the flow is smoother glide through the edge of airfoil and the worst model is the thickness adjusted model which the vortex of flow generate more drag. Again, the visual result is also hard to see but you can see more clearly when we change the velocity condition to match with the wind tunnel condition.

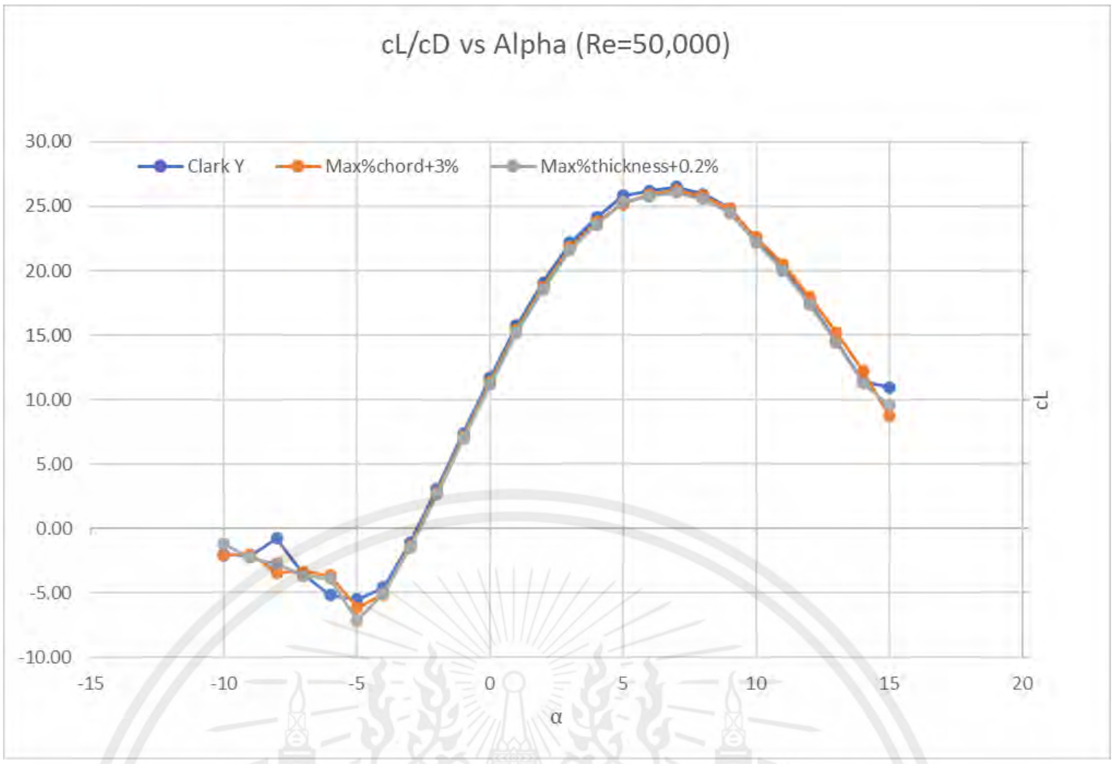


Figure 4.4 cL/cD of Clark Y and its mods

Lift to drag coefficient affects the performance of an airfoil when they are flying. In this case, The best performance airfoil is Clark Y since it generates more lift than the other models and results in less drag value. Only a few digits away from Chord adjusted models.

All of this result came from the condition that we can compare with the experimental result from resources that we have. So the next will be the result that we will compare with our own data from the wind tunnel that we use.

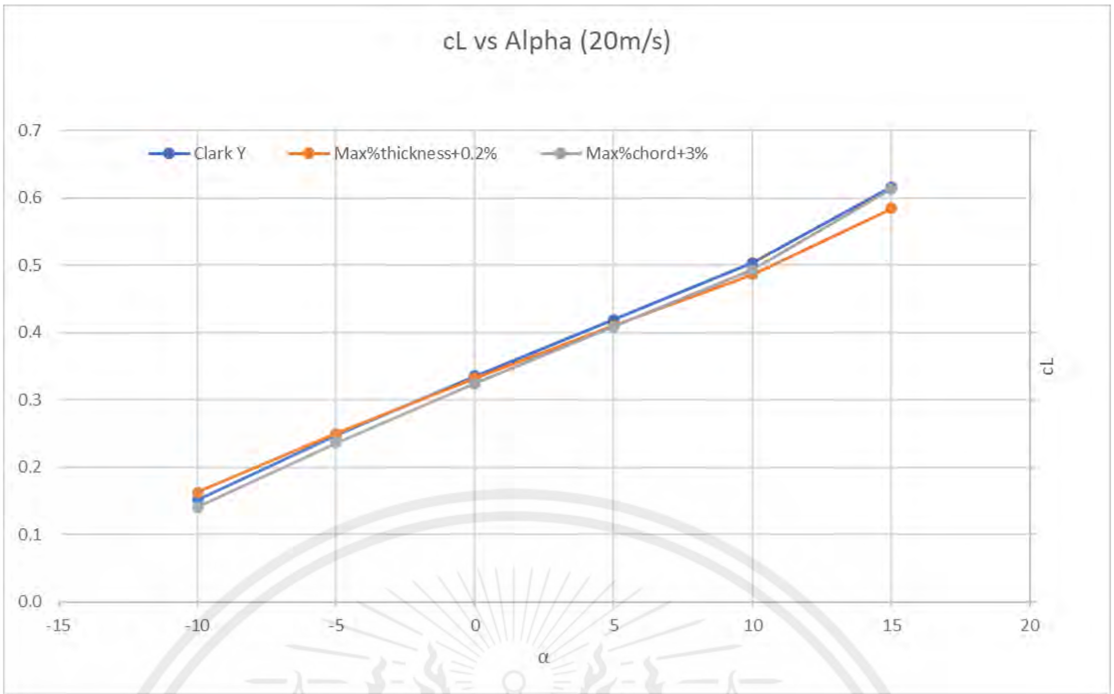


Figure 4.5 cL of Clark Y and mods at wind tunnel velocity

This condition we use 20 m/s from the range of wind tunnel air flow which is around 15 - 40 m/s. The result is greater in number of lift generated but trend is still the same which Clark Y model is greater at generating lift coefficient.

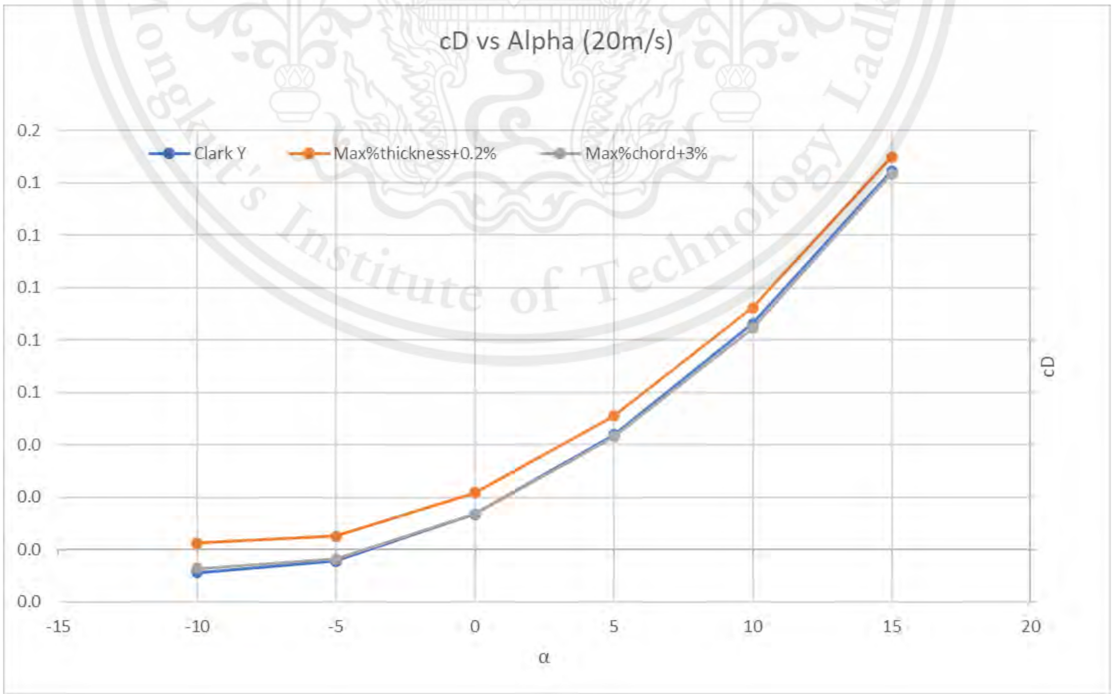
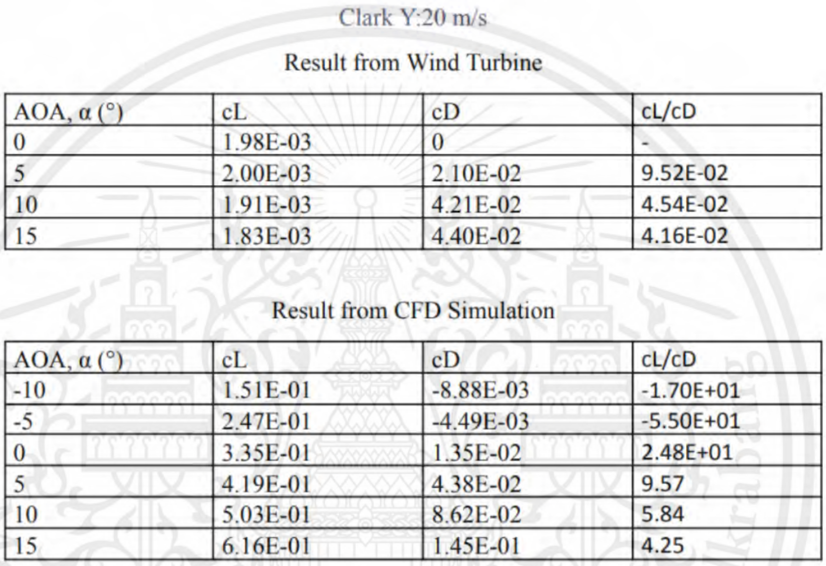


Figure 4.6 cD of Clark Y and mods at wind tunnel velocity

The drag coefficient now is more visible. The thickness model is clearly generated greater in drag. The turbulence causes more disturbance through the edge of this model so some of the flow didn’t attach to the edge which results in greater drag. The chord adjusted model could result in less drag if we move it further back because the flow delay makes the flow attach to the wall surface but still it generate less drag than Clark Y model in small amount.



Clark Y:20 m/s

Result from Wind Turbine

AOA, α (°)	cL	cD	cL/cD
0	1.98E-03	0	-
5	2.00E-03	2.10E-02	9.52E-02
10	1.91E-03	4.21E-02	4.54E-02
15	1.83E-03	4.40E-02	4.16E-02

Result from CFD Simulation

AOA, α (°)	cL	cD	cL/cD
-10	1.51E-01	-8.88E-03	-1.70E+01
-5	2.47E-01	-4.49E-03	-5.50E+01
0	3.35E-01	1.35E-02	2.48E+01
5	4.19E-01	4.38E-02	9.57
10	5.03E-01	8.62E-02	5.84
15	6.16E-01	1.45E-01	4.25

Figure 4.7 From top to bottom: Wind Tunnel and CFD Simulation results

The result from the wind turbine is not acceptable due to the values are clutter which is from the setup and condition of the machine. The lift and drag coefficient rise in contrast with the result that we have from simulation and experimental result from the resource that we reference. Therefore, we decide to focus and analyze the simulation that we compare with the resource that we have only.

4.2 Conclusion

In conclusion, the results from our CFD simulations and wind tunnel testing has proven that our attempt at modifying the Clark Y airfoil by increasing its maximum percentage of the chord by a factor of 3% for the same max camber and another increasing its thickness by a factor of 0.2% for the same chord in hopes of enhancing its lift-to-drag ratio has been ineffective. Even though in theory, the thickness of an airfoil is supposedly directly proportional to the airfoil's lift generation capability, that is not exactly the case. It is true that increasing the thickness of the airfoil can significantly impact its lift. A thick airfoil can generate more lift than a thin airfoil in a low subsonic speed as a thick airfoil means more surface area for the air to flow, and thus a greater pressure difference for superior lift generation. However, increasing the airfoil's thickness also increases its weight. The airfoil's mass and the gravity is what drags it down to earth. The Clark Y airfoil was designed with operations at low subsonic flow regimes in mind, thus, each feature and detail of the airfoil was intricately drawn to suit for those conditions. The airfoil's thickness and camber definition was already designed to be in balance. Tampering the design has upset the balance between the values of the features of the airfoil.

DISCUSSION

This section, we want to recommend setting up more conditions as we mention specialty organizations as they have more resources and background of the research through this topic. That will reduce errors and receive better results that approach more to the experimental result. SST k-ω model is easier to use and less time-consuming. However, the Generalized k-ω (GEKO) model takes more time to set, but if set correctly, would yield very accurate simulation results. We can narrow down the specification for research as this topic is specific. We can scope this to be the outcome or goal that we wanted this airfoil to work such as Higher lift at coefficient ability for slower speed flying or lower drag at high speeds and low lift coefficient rather than focusing on upgrade the airfoil only describing on lift and drag coefficient generated. That's our misunderstanding thinking about how we can upgrade or make the airfoil better. The objective should be clarified and clear before starting the experiment. This study helps us to understand some factors of the airfoil but also every factor is needed to be well calculated and instruct like we have some thought of adjusting them but some outcome didn't go as we thought it would be. For wind tunnel part, preparation on the machine need to be more precise and the machine needs to be inspect.

REFERENCES

[1] Anderson, J. D., & Cadou, C. (2024). *Fundamentals of aerodynamics*. McGraw-Hill.

[2] Henk Kaarle Versteeg, Weeratunge Malalasekera (2007). An Introduction to Computational Fluid Dynamics The Finite Volume Method-Pearson

[3] Beginner's Guide to Aerodynamics", Grc.nasa.gov, 2021.

[4] Turla, P. (2014). Retrieved from https://www.researchgate.net/publication/311562091_Aerodynamic_CFD_Study_of_Clark_Y_Airfoil

[5] Airfoil Tools. (n.d.). CLARK Y AIRFOIL (clarky-il). Retrieved from http://airfoiltools.com/airfoil/details?airfoil=clarky-il

[6] Edge, E. (n.d.). *Aerodynamics airfoil theory equations*. Engineers Edge - Engineering, Design and Manufacturing Solutions. https://www.engineersedge.com/fluid_flow/aerodynamics_airfoil_theory_equations__15301.htm

[7] Lazauskas, Leo Victor (2005), "Hydrodynamics of Advanced High-Speed Sealift Vessels", Master Thesis, University of Adelaide, Australia (https://digital.library.adelaide.edu.au/dspace/bitstream/2440/37729/1/02whole.pdf)

[8] Chopang. (2021). 63011190_Simulate air flow on wood/clark y airfoil in Ansys fluent. Retrieved from https://www.youtube.com/watch?app=desktop&v=8WsskqojKhI

[9] Thomas, C. (2023). NACA2412 tutorial in ansys fluent (student version) - lift, drag, angle of attack. Retrieved from https://youtu.be/3i9Ryq-m1HA?si=Q6X3jRqo6wdOZ9r4

[10] Jousefm. (2020). What is Y+ (yplus)? Retrieved from https://www.simscale.com/forum/t/what-is-y-yplus/82394

[11] Flow over an airfoil using ANSYS Workbench | Ansys courses. (n.d.). Retrieved from https://courses.ansys.com/index.php/courses/flow-over-an-airfoil/

[12] https://reddit.com/r/CFD/s/tEi0Pd2ccA

[13] Meshing in FEA: Structured vs unstructured meshes. (2020). Retrieved from https://onscale.com/meshing-in-fea-structured-vs-unstructured-meshes/

[14] What is CFD: What is Computational Fluid Dynamics? (2023). Retrieved from https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/

[15] Schlichting, Hermann (1979), Boundary Layer Theory, ISBN 0-07-055334-3, 7th Edition.

[16] Airfoil design. (n.d.). Retrieved from https://mail.tku.edu.tw/095980/airfoil%20design.pdf

[17] Prof. Van Buren. (2021). Aerodynamics. Retrieved from https://www.youtube.com/playlist?list=PLe2kb8k5FJpgLfhf18VphADpX9qvaby6k

[18] Lintermann, A. (n.d.). Retrieved from https://juser.fz-juelich.de/record/885994/files/2020_Lintermann_ClinBiomedEng_Ch6.pdf

[19] Summary of airfoil data - NASA technical reports server (NTRS). (1945). Retrieved from https://ntrs.nasa.gov/citations/19930090976

[20] IRA H. ABBOTT, ALBERT E. VON DOENHOFF, and LOUIS S. STIVERS, Jr. (1945). REPORT No. 824 SUMMARY OF AIRFOIL DATA . Retrieved from https://ntrs.nasa.gov/api/citations/19930090976/downloads/19930090976.pdf

[21] American Institute of Aeronautics and Astronautics. (1984). Retrieved from https://user.engineering.uiowa.edu/~me_160/CFD%20Labs/Lab2/clark-y-paper.pdf

[22] Silverstein, A. (1934). Scale Effect on Clark Y Airfoil Characteristics from NACA Full-Scale Wind-Tunnel Tests - NASA technical reports server (NTRS). Retrieved from https://ntrs.nasa.gov/citations/19930091575

[23] Piccirillo, A. (2000). The Clark Y Airfoil: A Historical Retrospective. Retrieved from https://www.semanticscholar.org/paper/The-Clark-Y-Airfoil:-A-Historical-Retrospective-Piccirillo/14d485fa945e1ea56fd6e975d090b4067d08b02a

[24] Turla, P. (2014). Aerodynamic CFD Study of Clark Y Airfoil. Retrieved from https://www.researchgate.net/publication/311562091_Aerodynamic_CFD_Study_of_Clark_Y_Airfoil

[25] Ansys. (2020). Retrieved from https://alfaproject.ir/wp-content/uploads/2020/04/part12.pdf

[26] Cadence. Retrieved from https://www.cadence.com/en_US/home/tools/system-analysis/computational-fluid-dynamics/y-plus.html

[27] Flying Staff. (2022, February 10). What Is Standard Temperature in Aviation? https://www.flyingmag.com/guides/what-is-standard-temperature-aviation/

[28] von Karman, Theodore (1934), "Turbulence and Skin Friction", J. of the Aeronautical Sciences, Vol. 1, No 1, 1934, pp. 1-20.

[29] Schlichting, Hermann (1979), *Boundary Layer Theory*, ISBN 0-07-055334-3, 7th Edition.

[30] Skin friction coefficient. (2016). Retrieved from https://www.cfd-online.com/Wiki/Skin_friction_coefficient

[31] Ahmed Al Makky. (n.d.). Number of Iterations. Retrieved from https://cfd2012.com/number-of-iterations.html

[32] Mazz. (2021). Clark Y airfoil Pros and Cons. Retrieved from http://www.thoracentesis.science/2021/02/Clark-Y-airfoil%20.html#google_vignette

[33] Fluid Mechanics for Mechanical Engineers/Technical Applications. (2024). Retrieved from https://en.wikiversity.org/wiki/Fluid_Mechanics_for_Mechanical_Engineers/Technical_Applications

[34] MIT Department of Aeronautics and Astronautics. (1997). Theory of Flight. Retrieved from https://web.mit.edu/16.00/www/aec/flight.html

[35] Baum, C. (2022). How the NACA 2412 Aerofoil Saves You. Retrieved from https://www.absolutepilots.com/en/how-the-naca-2412-aerofoil-saves-you-e/

[36] Baals, D. D., & Corliss, W. R. (1981). Wind Tunnels of NASA - NASA Technical Reports Server (NTRS). Retrieved from https://ntrs.nasa.gov/citations/19810023610

[37] Bowers, A. (2011). Laminar Flow and the Holy Grail. Retrieved from https://blogs.nasa.gov/armstrong/2011/02/11/post_1296777084480/

[38] The Efficient Engineer . (2020). Understanding Bernoulli's Equation. Retrieved from https://www.youtube.com/watch?v=DW4rItB20h4

[39] The Efficient Engineer. (2021). Understanding Aerodynamic Drag. Retrieved from https://www.youtube.com/watch?v=GMmNKUlXXDs

[40] The Efficient Engineer. (2020). Understanding Laminar and Turbulent Flow. Retrieved from https://www.youtube.com/watch?v=9A-uUG0WR0w

[41] Dough, J. (2022). Airfoil Design 101: What Is an Airfoil? Retrieved from https://www.naa.edu/airfoil-design/

[42] | RTAF History. (n.d.). Retrieved from https://rtaf3en.rtaf.mi.th/history

[43] Lednicer, D. (2023). The Incomplete Guide to Airfoil Usage. Retrieved from https://m-selig.ae.illinois.edu/ads/aircraft.html

[43] Selig, M. (2024). UIUC Airfoil Coordinates Database. Retrieved from https://m-selig.ae.illinois.edu/ads/coord_database.html

[44] Air Shaper, W. R. (2020). CFD Results - How to Interpret an Aerodynamic Analysis. Retrieved from https://www.youtube.com/watch?v=XntO27S5Urc

[45] Wimshurst, A. (2019). [CFD] The k - omega SST Turbulence Model. Retrieved from https://www.youtube.com/watch?v=myv-ityFnS4

[46] Aerolab. (2021). Open Circuit Wind Tunnels . Retrieved from https://www.aerolab.com/aerolab-products/open-circuit-wind-tunnels/

[47] Rahul. (2016). modifying the airfoil in XFLR5. Retrieved from https://www.youtube.com/watch?v=t4X-nW_utK0

[48] Rahul. (2016a). How to modify thickness and camber of airfoil using profili. Retrieved from https://www.youtube.com/watch?v=jSyU2bwMtm0

[49] https://neal.fun/space-elevator/

AUTHOR BIOGRAPHY

Author: Mr. Napath Hankampa
Degree: Mechanical Engineering
Date: 27 October 2023
Date of Birth: 8 June 2003
Place of Birth: Nakhon Sawan, Thailand

Undergraduate and Graduate Education:

Bachelor degree in Mechanical Engineering,
King Mongkut’s Institute of Technology Ladkrabang, Bangkok, 2024

Major: Mechanical Engineering

AUTHOR BIOGRAPHY

Author: Mr. Kannawat Vayuwattanasiri
Degree: Mechanical Engineering
Date: 27 October 2023
Date of Birth: 12 September 2002
Place of Birth: Rayong, Thailand

Undergraduate and Graduate Education:

Bachelor degree in Mechanical Engineering,
King Mongkut’s Institute of Technology Ladkrabang, Bangkok, 2024

Major: Mechanical Engineering