FLOW ANALYSIS OF AIRFOIL WITH MECHANICAL SLAT AND FLAP USING CFD

CHAN CHUN HONG

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

FLOW ANALYSIS OF AIRFOIL WITH MECHANICAL SLAT AND FLAP USING CFD

CHAN CHUN HONG

This report is submitted
in fulfilment for the degree of
Bachelor of Mechanical Engineering

FACULTY OF MECHANICAL ENGINEERING

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

JULY 2021

DECLARATION

I declare that this project report entitled “Flow analysis of airfoil with mechanical slat and flap using CFD” is the result of my own work except as cited in the references

Signature : ....

Name : ....

Date : ....

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature : ....

Name : ....

Date : ....

:

DEDICATION

To my beloved mother and father

ABSTRACT

Aeroplane have increasingly become an indispensable means of transportation in modern civilization. The design of the airfoil such as slat and flap will affect the fuel consumption and the performance of the aeroplane. Therefore, obtaining a good slat and flap angle of airfoil could increase the performance of aeroplane. In the present work, the numerical analysis lift performance of NACA 0015 airfoil in low Reynold numbers from angle of attack, 0° to 20° has been carried out. The present study aims to improve the lift coefficient of NACA 0015 airfoil. The mechanical slat and flap is designed in order to improve the lift coefficient of airfoil when angle of attack, α= 17°. An optimization of the slat and flap angle is required for design the effective mechanical slat and flap. For NACA 0015 base case with the mechanical slat and flap, there were 3 set of setting, (δ = 15°, β = 25°), (δ = 20°, β = 30°) and (δ = 25°, β = 35°) have been conducted in the simulations. The simulation was performed using CFD program in Ansys Fluent software. The numerical results of lift coefficient, contour of velocity and static pressure around the airfoil have been determined and compared with the results obtained from NACA 0015 airfoil. As a result, the stall angle for NACA 0015 base case and airfoil with mechanical slat and flap is at 17° when air velocity at 10 m/s. Next, the NACA 0015 base case applied the mechanical slat and flap will increase the lift coefficient from at least 62% to 110%. In the simulations, the airfoil with mechanical slat at 20° and flap at 30° obtained the maximum lift coefficient which was 0.3107. Indirectly, the application of slat and flap could increase the critical angle of attack of the airfoil and the area contact between the air and the bottom surface of airfoil. In future, it is recommended that the setting of input and boundary conditions should be applied based on the requirement carefully.

ABSTRAK

Kapal terbang semakin menjadi alat pengangkutan yang sangat diperlukan dalam peradaban moden. Reka bentuk kerajang udara seperti selat dan kepak akan mempengaruhi penggunaan bahan bakar dan prestasi kapal terbang. Oleh itu, memperoleh sudut selat dan kepak yang baik bagi kerajang udara dapat meningkatkan prestasi kapal terbang. Dalam karya ini, analisis angka meningkatkan prestasi NACA 0015 dalam jumlah rendah Reynold dari sudut serang, 0° hingga 20° telah dilakukan. Kajian ini bertujuan untuk meningkatkan pekali angkat udara NACA 0015. Selat dan kepak mekanikal dirancang untuk meningkatkan pekali angkat udara semasa sudut serangan, α = 17°. Pengoptimuman sudut selat dan kepak diperlukan untuk merancang selat dan kepak mekanikal yang berkesan. Untuk kes asas NACA 0015 dengan selat mekanik dan kepak, terdapat 3 set tetapan, (δ = 15°, β = 25°), (δ = 20°, β = 30°) dan (δ = 25°, β = 35°) telah dijalankan dalam simulasi. Simulasi dilakukan menggunakan program CFD dalam perisian Ansys Fluent. Hasil berangka dari pekali angkat, kontur halaju dan tekanan statik di sekitar kerajang udara telah ditentukan dan dibandingkan dengan hasil yang diperoleh dari NACA 0015. Hasilnya, sudut tegun untuk kes asas NACA 0015 dan kerajang udara dengan selat dan kepak mekanikal berada pada 17° apabila halaju udara pada 10 m/s. Seterusnya, kes asas NACA 0015 yang digunakan selat mekanikal dan kepak akan meningkatkan pekali angkat dari sekurang-kurangnya 62 % hingga 110%. Dalam simulasi, kerajang udara dengan selat mekanikal pada 20° dan kepak pada 30° memperoleh pekali angkat maksimum iaitu 0.3107. Secara tidak langsung, penerapan selat dan kepak dapat meningkatkan sudut serang kritikal bagi kerajang udara dan kontak kawasan antara udara dan permukaan bawah kerajang udara. Di masa depan, disarankan agar pengaturan syarat input dan batas harus diterapkan dengan teliti berdasarkan keperluan.

ACKNOWLEDGEMENTS

First of all, I would like to express my greatest gratitude to my supervisor Ir Dr Nazri Bin Md Daud from Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka in guiding me throughout the process of research. I am immensely grateful to him for sharing his knowledge in his expertise field and encouragement throughout the duration of this project.

In addition, I would like to thank my parents Mr. Chan Foon Man and Mrs. Lim Fee Chu for their constant support, encouragement and recognition to me.

Last but not least, I would like to thank my classmates Gok Wei Keong and Ooi Jun Wai for guiding me through the final year project. I am grateful to have met a group of friends who have always been supportive of me during my time as a university student.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

A	airfoil cross-sectional area, Van Driest damping constant
A'	Roe-averaged flux vector Jacobian matrix associated with Ƒ
B'	Roe-averaged flux vector Jacobian matrix associated with Ǵ
a	speed of sound
ai, a_2	cell face areas
b	wake width
c	airfoil chord
Ca	axial force coefficient
C_d	drag coefficient
C_l	lift coefficient
Cm	quarter-chord moment coefficient
Cn	normal force coefficient
D	$D^+ - D^-$
$D^\pm$	portion of A associated with positive/negative eigenvalues
dF	MINMOD of flux difference
E	$E^+ - E^-$
$E^\pm$	portion of B associated with positive/negative eigenvalues
Et	total energy
e	specific internal energy

F, G	flux vectors
H	boundary layer shape factor
h	enthalpy, representative cell width
I	identity matrix
I^*	boundary conditions transformation matrix
J	Jacobian
J^*	$J\,I^*$
K	law of the wall constant
M	Mach number
N	viscous Jacobian matrix
n	time index
p	pressure
Pr	Prandtl number
Prt	turbulent Prandtl number
Q	vector of flow variables
q_x, q_y	heat flux in x and y directions
R	matrix of right eigenvectors
R^{-1}	matrix of left eigenvectors
R^{*-1}	matrix of left eigenvectors with zeroed rows
ρ	density
σ	time integration safety factor
R_e	Reynolds number based on airfoil chord
r	nose radius
T	temperature
t	time
U	contravariant velocity in ξ direction

Ue	boundary layer edge velocity
u	velocity component in the x direction
u_r	friction velocity
V	contravariant velocity in the r_j direction
v	velocity component in the y direction
w	wake velocity defect
x,y	coordinates in physical space
Xt	distance downstream from trailing edge
y^+	nondimensionalized boundary layer distance
Z	inverse of boundary conditions matri
α	angle of attack
s	boundary layer thickness
6'	boundary layer displacement
Ө	weighting of flux
Ө	time boundary layer momentum thickness
k	modified airfoil area
u	coefficient of molecular viscosity
$^u t$	coefficient of eddy viscosity
v	coefficient of kinematic viscosity
$^v t$	coefficient of kinematic eddy viscosity

CHAPTER 1

INTRODUCTION

1.1 Background

Aerodynamics is the study of how gases interrelate with moving bodies and usually encounter is air. Firstly, aerodynamics is related to drag and lift forces, which are caused by air flowing over and around solids body. Nowadays, aerodynamics was used to design products, building or things relate to pressure concern, including Formula 1 racing sport, bridges and train. For example, a fireworks explodes from the ground, a kite able to fly and aerodynamics even affect the cars. (Lucas, 2014) However, engineers are most concerned about the aerodynamics of aircraft and automobiles. This is because aerodynamics is the fundamental science for explaining aircraft concepts, and airfoil is the main part and most important structure to produce lift and drag forces. Therefore, the rules of aerodynamics can explain how aeroplanes fly.

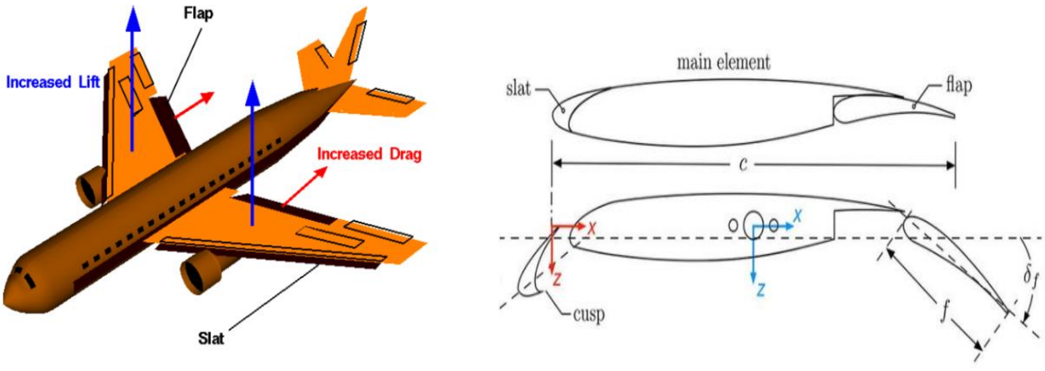
The appearance on airfoil is the flow of fluid impose on the object becomes a crucial issue in devising an aeroplane, especially the cap is fluid flowing around airfoil. From mechanical point of view, the lift generated by the airfoil depends on the shape of it and its area, aircraft speed and position. A larger wing area is required for aircraft taking off and landing. Slat is an additional area that be installed in front of the leading edge while the component behind the trailing edge called flap. The slats and flaps move along the chord to increase the area of the wing by changing the angle or adjusting the distance. Several types of flaps can be used, depending on the type of aircraft, weight, runway conditions and so on. Mainly Plain Flap, Fowler Flap, Slotted Flap, Split Flap, Zap Flap, Double Slotted Flap, Junkers Flap, Gouge Flap and Krueger Flap.(Sarjito et al., 2017)

Computational fluid dynamics (CFD) is under of fluid mechanics. The field of computational fluid dynamics has become a commonly used tool for generating fluid flow solutions with or without solid interactions.(SimScale, 2017) Numerical analysis and data structures are usually used to analysis and solve problems related to fluid flow. For analysis problems involves to fluid flow, computers are used to execute the calculations to simulate the free-stream flow of the fluid or air, and the interplay of the fluid (liquids and gases) with surfaces defined by boundary conditions. Computational fluid dynamics (CFD) really helps to replace experimental averages. There are some identical software like Ansys, OpenFOAM, PowerFLOW, SimScale and Autodesk CFD could solving the fluid flow in specific. From that, it can use the governing equation for discretization and iteratively solve each control variable to get the approximate value of each variable in the computational domain. In this study, Ansys Fluent software was be chosen this is due to the reason that this software can solve most sophisticated models for multiphase flows, reacting flow, fluid-structure interaction and even complicated viscous and turbulent, internal and external flows. In this project, relationship between flap aerodynamics and slats coefficient of the NACA 0015 airfoil was investigated.

1.2 Problem Statement

Generally, the aerodynamic structure of the aircraft will affects the speed and fuel consumption of the flight. Figure 1.1 shows airfoil of the aircraft during flight. A bad design of airfoil will consumes a lot of fuel without increasing speed during flight. Moreover, a bad slat and flap angle of the airfoil will lead to inefficient for departure and most serious is that it will make the aircraft dangerous. Therefore, obtaining a good slat and flap degree could greatly increase the lift force and reduce fuel consumption. For example, an over lift coefficient will give faster take off but it will also lead to more drag force on the aircraft and make it crash. Figure 1.2(a) shows the effect of slats and flaps on lift and drag forces. The slats and flaps airfoil will definitely be put away during flight. Therefore, obtaining a good slat and flap angle of airfoil was the main mission of this project since it can plainly make the aeroplane have suitable lift coefficient. Figure 1.2(b) shows the opening and closing the mechanical slat and flap.

Figure 1.1: Airfoil of the aircraft during flight.



(a) (b)

Figure 1.2 (a): Effect of slats and flaps on lift and drag forces.

(b): Opening and closing the mechanical slat and flap.

1.3 Objective

The objectives of this project are as follows:

1. To design airfoil model NACA 0015 with mechanical slat and flap.

2. To determine and compared the lift coefficient between base case airfoil and airfoil with mechanical slat and flap.

1.4 Scope of Project

The scopes of this project are:

1. Base case of model NACA 0015 airfoil and added slat flap angle airfoil will be created by using SolidWorks, Ansys Fluent software or other drawing software.
2. The chord length has been set to 100 mm, angle of attack, $\alpha = 17°$, velocity magnitude: 10 m/s.
3. The result of simulations of fluid flow around the airfoil.
4. The lift coefficient created by different types of airfoil.

1.5 General Methodology

Listed below are the actions required to achieve the project's goals.

1. Literature review

   Journals, articles and any materials related to the project will be reviewed.

2. Inspection

   NACA 0015 airfoil will be referenced to identify coordinates shape of airfoil and directly used it for adding slat and flap angle airfoil on it.

3. Simulation

   Simulation of the fluid flow around the airfoil will be made based on the types of airfoils.

4. Analysis and suggested solution

   Analysis will be presented on how the designed slat and flap angle airfoil will affect the lift coefficient of aircraft. Solutions will be proposed based on the analysis.

5. Report writing

   A report on this study will be written at the end of the project.

CHAPTER 2

LITERATURE REVIEW

2.1 Deflection Airfoil Flap Angle Using CFD Simulation Techniques

A research CFD Analysis of airfoils with flaps for low Reynold numbers (Fonseca et al., 2019) also states that there had a superior lift coefficient (CL) and drag coefficients (CD) similar to those provided by the symmetrical profile when only the airfoils with flap are analysed. High lift gadgets such as flaps and slats can increase the lift coefficient on an airfoil. An analysis of the airfoil flap deflection angle using CFD simulation techniques (Thejaraju et al., 2019) stated that: "The coefficients of lift and drag increase to a certain point as the flap deflection angle increases. However, beyond a certain angle, CL drops." Figure 2.1 shows the Comparison between Exp. & CFD results. After comparing airfoil CFD simulations taken from the online airfoil plotter and the fabricated model tested in the wind tunnel. It can be inferred that there is a minimum difference in the values between CFD and experimental outcomes by analysing the graph plotted experiment and CFD outcomes. In reality, in many tests, this CFD simulation or test method can save a lot of time, especially without creating a true model airfoil. Therefore, since the error is small, CFD validation can be used to obtain the optimised performance. Figure 2.2 shows the lift coefficient at stall point with and without flap angle of the airfoil. The equation of lift and drag coefficient was provided.