Application of biologically inspired "Pop-Up" feather style high lift device on micro aerial vehicles.

Wang, John Chung Hung.

2013

Wang, J. C. H. (2013). Application of biologically inspired "Pop-Up" feather style high lift device on micro aerial vehicles. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/55002>

<https://doi.org/10.32657/10356/55002>

NANYANG
TECHNOLOGICAL
UNIVERSITY

**Application of Biologically Inspired
“Pop-Up” Feather Style High Lift Device on
Micro Aerial Vehicles**

Author:

Chung Hung John Wang

Supervisor:

Asst. Professor Jörg Schlüter

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in fulfillment of the requirement for the degree of
Doctor of Philosophy

2012

Abstract

Recent years saw the increase in relevance of Micro Aerial Vehicle (MAV) in both civilian and military applications. One of the more attractive feature of MAV is the lower cost and ease of transport due to its smaller size; however, the size and weight of the MAV also limited the forms of high lift device that could be applied to the vehicle to increase its operating envelope. One of the devices proposed is the “Pop-Up” feather type passive high lift device (passive flap), a design based on observation of bird feathers during landing. The primary objective of the current research is to investigate the performance of passive flap when applied to the lower operating Reynolds number of MAVs ($Re = 40,000$) and establish the CFD and experimental procedure for optimization of passive flap design for application on MAV. To that end, two simulation procedures were developed: one using steady-state solver and optimization algorithm to seek the equilibrium flap angle; the other directly solves for the movement of the flap based on forces acting on the flap and the flap inertia. The solutions from the resulting solvers agrees favorably with existing water tunnel experimental lift data for 2D airfoil with and without passive flap, and while the solutions were not fully validated for detailed flow features, they could be used to supplement the experimental data to better evaluate the impact of different flap size (c_f) and flap position (x_f) on the performance of the flap under 2D settings. On the other hand, it was recognized that three-dimensional flow features could have major impact on the performance of the flap, namely the wing-tip vortex which alters the effective angle of attack along the span of the wing. As no prior studies on the application of passive flap on finite wing existed, wind tunnel studies were performed at $Re = 40,000$ on a rectangular wing model with varying size of flaps spanning from the center-line of the wing (b_f); finite wing simulations were also performed to obtain visualization on flow structure around the wing, though further validation of the 3D results needs to be left for future work. It should be noted that there remains outstanding issues to be addressed in the future. Some anomalous results show that the numerical discretization requires a body fitted mesh in the near wall region and further validation work is required to assert accuracy. While some mean aerodynamic characteristics may agree well between the numerical and experimental results, the computed detailed flow behavior along the airfoil surface regions will require further validation. In particular, the experimental diagnostics were only limited to measurements of the integrated quantities of lift.

Acknowledgements

Special thanks to Defense Science Organization for funding the current project, Professor Schlüter for his guidance throughout my research, and all my fellow PhD students in the Aerodynamics Lab for their ideas and suggestions.

Contents

Contents	I
List of Figures	III
List of Tables	VI
Nomenclature	VII
1 Introduction	1
2 Theory and Related Studies	7
2.1 Theoretical Background	7
2.2 Studies Prior to Current Project	8
2.3 Post-2008 Studies	15
3 Experimental Setup	17
3.1 Overview of Wind Tunnel Facility	17
3.2 Rectangular Wing Model	20
3.2.1 Manufacturing of Wing	20
3.2.2 Passive Flap Setup	23
3.3 Adaption on a Remote-Control Aircraft	26
3.3.1 Aircraft Model	26
3.3.2 Passive Flap Configurations	27
4 Flow Solvers and Numerical Methods	29
4.1 Simulation Strategies	29
4.1.1 Steady-State Simulation Based Method	30
4.1.2 Transient Simulation Based Method	31
4.2 Overview of OpenFOAM	31
4.2.1 Development of OpenFOAM	32
4.2.2 Implementation of Navier-Stokes [1]	33
4.2.3 Turbulence Modeling	39

4.2.4	Handling of Flap Motion	47
4.3	Development of Solvers	54
4.3.1	The Steady-State Solver	54
4.3.2	Transient Fluid-Induced-Motion Solver	56
5	Simulation Setup	61
5.1	2D Simulation Setup	62
5.1.1	Steady-State Simulation Setup	62
5.1.2	Transient Simulation Setup	66
5.2	3D Simulation Setup	72
6	Airfoil Simulation Studies	77
6.1	Steady-State Simulation	78
6.1.1	$x_f/c = 0.8$, $c_f/c = 0.2$ Flap Configuration	79
6.1.2	$x_f/c = 0.6$, $c_f/c = 0.4$ Flap Configuration	85
6.1.3	Discussion	91
6.2	Transient Simulation	96
6.2.1	Simulation Results	96
6.2.2	Discussion	103
7	Finite Wing Studies	108
7.1	Experimental Results	108
7.1.1	The Effect of Flap Span b_f	109
7.1.2	The Effect of Flap Position x_f	116
7.1.3	The Effect of Flap Chord c_f	118
7.1.4	The Effect of Reynolds Number	120
7.1.5	Drag Data	123
7.1.6	Full Model Results	124
7.2	Simulation Results	126
7.2.1	Finite Wing Simulation Results	128
7.3	Comparison and Discussion	135
8	Conclusion	138
8.1	Achievements	139
8.2	Future Work	140
	Bibliography	142

List of Figures

1.1	The Ryan Firebee Drones	2
1.2	RQ-11 and RQ-1 Command Center Compared	3
1.3	Cl_{max} and Reynolds Number	4
1.4	Lifted feather of Bird	5
2.1	Flow Field with and without Passive Flap	8
2.2	Passive flap setup for glider flight test	11
2.3	Lift Curve from Kernstine et al.	14
2.4	Lift Curve from the Schlter Study	16
3.1	NTU Wind Tunnel System	19
3.2	Aluminum Wing Mount used in the Experiment	20
3.3	SD8020 Balsa Wing Model	21
3.4	Accuracy of the Model	22
3.5	Flap Material for Rectangular Wing	23
3.6	Flap Configurations Tested on the Rectangular Wing	25
3.7	Multiplex Merlin Model in Wind Tunnel	27
3.8	Flap Material For RC Model Experiments	28
4.1	Strategy for Steady-State Simulation	30
4.2	Turbulence Models Comparison	45
4.3	snappyHexMesh Meshing Process, Overall	49
4.4	snappyHexMesh Meshing Process, Detailed	50
4.5	Diffusivity Study	52
4.6	Diffusivity Study	53
4.7	Equilibrium Angle Seeking Process	56
5.1	Steady-State Model and Boundary	63
5.2	Steady-State Mesh	64
5.3	Steady State Mesh Convergence Study	65
5.4	Unsteady Model and Boundary	67
5.5	Transient Coarse Mesh	69

5.6	Transient Dense Mesh	70
5.7	3D Wing Model	74
5.8	3D Cross Section	75
5.9	3D Z Cross Section	76
6.1	Lift Curve for SD8020	78
6.2	Lift Curve for $x_f/c = 0.8$, $c_f/c = 0.2$	79
6.3	Velocity Contour $c_f/c = 0.2$; $x_f/c = 0.8$	80
6.4	Velocity Contour $c_f/c = 0.2$; $x_f/c = 0.8$	81
6.5	Velocity Contour $c_f/c = 0.2$; $x_f/c = 0.8$	82
6.6	Velocity Contour $c_f/c = 0.2$; $x_f/c = 0.8$	83
6.7	Velocity Contour $c_f/c = 0.2$; $x_f/c = 0.8$	84
6.8	Lift Curve for $x_f/c = 0.6$, $c_f/c = 0.4$	85
6.9	Velocity Contour $c_f/c = 0.4$; $x_f/c = 0.6$	86
6.10	Velocity Contour $c_f/c = 0.4$; $x_f/c = 0.6$	87
6.11	Velocity Contour $c_f/c = 0.4$; $x_f/c = 0.6$	88
6.12	Velocity Contour $c_f/c = 0.4$; $x_f/c = 0.6$	89
6.13	Velocity Contour $c_f/c = 0.4$; $x_f/c = 0.6$	90
6.14	Stream Trace Ribbon Comparison: Recirculation	92
6.15	Stream Trace Ribbon Comparison: Free Stream	93
6.16	Imperfection with SnappyHexMesh	94
6.17	Steady-State Boundary Grid and Boundary Layer Prediction	95
6.18	Lift Curves From 2D Transient Simulations	97
6.19	Transient Velocity Solutions Comparison	99
6.20	Transient Pressure Solutions Comparison	100
6.21	Recirculation Flow Example 1	101
6.22	Recirculation Flow Example 2	102
6.23	Unstable k and ω	105
6.24	Lift over Drag Ration Plot	106
6.25	Moment Coefficient	107
7.1	Flap span study 1	110
7.2	Flap span study 2	111
7.3	Flap span study 3	112
7.4	Flap span study 4	113
7.5	Flap span study 5	114
7.6	Flap span study 6	115
7.7	Lift curves of $c_f/c = 0.3$ passive flaps at different chord-wise position at $Re = 4 \times 10^4$	117

7.8	Lift curves of $0.8b_f/b$ passive flaps with various flap chord-length at $Re = 4 \times 10^4$ compared with data without the flap. The flaps are positioned at $0.7x_f/c$ for 7.8(a) and $0.6x_f/c$ for 7.8(b).	119
7.9	Reynolds Number Effect 1	121
7.10	Reynolds Number Effect 2	122
7.11	Finite Wing Drag Data	123
7.12	The lift curve of the aircraft mode with and without the passive flap device installed at $Re = 40,000$ and $Re = 60,000$. .	125
7.13	Ribbon Plot for 3D Test Case	127
7.14	3D Velocity and Pressure Profiles 1	129
7.15	3D Velocity and Pressure Profiles 2	130
7.16	3D Stream Traces	131
7.17	URANS and LES Results Comparison 1	133
7.18	URANS and LES Results Comparison 2	134
7.19	Pressure Contour on Flap Surface	136

List of Tables

3.1	Specification of the NTU Wind Tunnel System	18
4.1	Constants for Spalart-Allmaras Turbulence Model	42
4.2	Constants for k - ω SST Turbulence Model	44
5.1	Fluid Property	61
5.2	Boundary Conditions	65
5.3	Initial Conditions	66
5.4	Cell Counts and Dimensionless Wall Distance for Fine and Coarse Meshes	68

Nomenclature

α	Angle of attack, (degree).
β	Passive flap deployment angle, (degree).
ρ_∞	Free stream fluid density (kg/m^3).
b	Wing span (m).
b_f	Span-wise size of the passive flap (m).
b_w	Distance between the root of the wing to the tip (m).
c	Chord of the airfoil (m).
c_f	Length of flap in the chord-wise direction (m).
c_l, c_a	Sectional lift coefficient, $c_l = \frac{l}{\frac{1}{2}\rho_\infty U_\infty c}$.
c_m	Coefficient of moment, $c_m = \frac{m}{\frac{1}{2}\rho_\infty U_\infty S c}$
c_p	Pressure coefficient, $c_p = \frac{p-p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$
l	Sectional lift force (N/m).
m	Pitching moment on the airfoil (Nm).
p_∞	Free stream pressure (Pa).
S	Wing area (m^2).
U_∞	Free stream flow velocity (m/s).
x_f	Chord-wise position of the flap hinge measured from the leading edge of the airfoil (m).

Chapter 1

Introduction

The concept of using unmanned aerial platform for weapon delivery has been around as early as mid 1800s, when the Austrian army besieging Venice launched balloons carrying remotely released bombs [2]; though the device itself cannot be strictly categorized as unmanned aerial vehicles by the modern definition, it did open a new path for future weapon developments. The modern form of Unmanned Aerial Vehicles (UAV), as defined by the U.S. Department of Defense “A powered, aerial vehicle that does not carry a human operator, uses aerodynamic forces to provide vehicle lift, can fly autonomously or be piloted remotely, can be expendable or recoverable, and can carry a lethal or nonlethal payload. Ballistic or semi-ballistic vehicles, cruise missiles, and artillery projectiles are not considered unmanned aerial vehicles, [3] ” was first developed in the 1960s to replace the increasingly vulnerable U-2 reconnaissance aircraft.

Conducting reconnaissance missions with UAVs has several advantages over that with conventional aircraft: the lack of risk to human life; the re-

duced risk of losing sensitive information; the increased mission duration due to the removal of life support system (reduction in weight), and the ability of having multiple staff operating the aircraft in shifts. The first UAV to served in a war was the Ryan Model 174 Lightning Bug (Figure 1.1), a modified version of the unmanned target drone Ryan Firebee, with the mission of monitoring People's Liberation Army for possible Chinese engagement in the Vietnam War. Though not as successful as originally hoped due to the inherited inaccuracy of dead reckoning system, the Ryan Lightning Bug series of UAV went on to perform thousands of mission until the end of Vietnam War [4]. The study of using UAV for offensive mission began in 1971, when four Ryan Model 174S drones were converted to include weapon pylons and direct TV data link; although these early Unmanned Combat Aerial Vehicle (UCAV) worked well under test conditions, the weakness from jamming of data link limited their use in the battlefield [5].

Figure 1.1: Derivatives of the Ryan Firebee reconnaissance drone. [4]

The development of global positioning system removed the uncertainty in navigation associated with gyro-based dead reckoning system, and allowed UAV to perform more precised missions. The refinement of remote sensing

technology also enabled UAV to gather intelligence with higher accuracy and much improved image resolution, with the ability to penetrate clouds and sandstorm. The advancement in navigational aid, sensor suite, and weapon carrying capability lead to the development of high altitude long endurance reconnaissance aircraft like the *RQ – 4* Global Hawk and *RQ – 1/MQ – 1* Predator, both of which saw extensive action in the recent war in Afghanistan.

Though highly capable as intelligence gathering platform, the size of the UAV as well as that of the command and control infrastructure prevented the downward distribution of this technology, and discouraged a more wide spread usage of the UAV in both military and civilian applications. Luckily, recent advancement in miniaturization of electronic sensors and improvement in battery technology makes the development of Micro Aerial Vehicle (MAV) a feasible option. The command and control infrastructure required for MAV and UAV is compared in Figure 1.2.

(a) RQ-11 Raven [6]

(b) RQ-1 Predator [7]

Figure 1.2: Command and control facility for RQ-11 Raven, a MAV, compared to that of RQ-1 Predator.

Though less complex and less capable comparing to UAV, MAV is gen-

erally cheaper, easier to operate and maintain, and allows for more direct control over what information to gather and where. With its small size and relatively low cost, MAV can be deployed down to the smallest field unit inside a combat zone. The cost could further decrease when used in civilian operation as less sophisticated data encryption will be required, allowing wide spread use of the MAV for search and rescue operation, wild life tracking, traffic monitoring, and even help in police manhunt.

One major design challenge faced by MAV designers is the Reynolds number effect: the phenomenon that as the size and speed of the aircraft decreases, so does the maximum lift coefficient for a particular aerodynamic design. NACA scientists Furlong and Fitzpatric [8] summarized the effect of Reynolds Number on lift as follows, “The effect of increasing Reynolds number is to cause an earlier transition from laminar to turbulent boundary layer. The increased turbulent boundary layer is then capable of resisting separation, and a higher angle of attack is reached before stalling occurs; thus, an increase in maximum lift coefficient is obtained.”

Figure 1.3: Maximum lift coefficient as a function of Reynolds Number [8].

The usual approach to increase aircraft lift generation, which will also

expand the flight envelope and increase aircraft maneuverability, is the installation of high lift device. Active devices such as flap and slat alters the camber of the airfoil at high angle of attack, delaying flow separation and producing the additional lift required for the flight maneuver. The selection of high lift devices suitable for MAV, however, is very limited, as the size and weight limitation of the aircraft ruled out most of the tried and tested devices. Fortunately, biological examples that inspired human flight provided yet another natural contraption that has the potential of providing light and easy to install high lift device: observation of bird's secondary-covert feather during take-off and landing lead engineers to believe that the pop-upped feather forms a natural flow control device that helped increase the lift generated by the wing (Figure 1.4).

Figure 1.4: Lifted covert feather of a bird landing [9].

The primary objective of the current PhD thesis study is to perform applied aerodynamic study aiming to evaluate the flow physics effecting the

performance of the “pop-up feather” type high lift device ¹ at the relatively low Reynolds number of 40,000 as well as the design criteria through experimental and computational studies. The hope is to provide a usable guide-line as well as a CFD tool set that can be used to aid the design process of MAV designer who wanted to incorporate the passive device onto their aircraft.

¹Referred to in the main text as passive flap

Chapter 2

Theory and Related Studies

2.1 Theoretical Background

The earliest documentation that mentioned using a feather-like passive device for upper airfoil flow control was an article in the German “Aerokurier” magazine [10] that described the experiments done at the German Aerospace Center (DLR) during the Second World War. The idea behind the device was to utilize the recirculation flow within the laminar separation bubble to lift up the passive flap in a similar fashion to the lifted feather in Figure 1.4. The the passive flap maintains its elevated position from the pressure difference across the flap, and would alter the upper surface profile of the wing, encouraging the reattachment of the flow and preventing the formation of turbulence flow separation behind the wing. The reduction in the scale of turbulence separation makes the drop in lift generation at high angle of attack less drastic, while delaying the formation of laminar separation bubble at the original stall angle. Figure 2.1 illustrates the effect of the passive flap

on the wake behind an airfoil.

Figure 2.1: Illustration of flow fields behind an airfoil without (left) and with the passive flap device.

The first systematic study of the passive flap was published in 1996 by Patone and Müller [11] of Technische Universität Berlin with funding from the DLR; this study and the subsequent studies on the passive flap are documented in the following sections.

2.2 Studies Prior to Current Project

The Patone and Müller study [11], as mentioned in the previous section, was the first comprehensive wind tunnel study of the passive flap device. The study was conducted at $Re = 1.3 \times 10^5$ using a NACA2412 airfoil model with a 4 : 3 aspect ratio as the test platform, and the material for the passive flap was chosen to mimic the flexibility and porosity of the covert feather: spring steel wires were used to simulate the quills and silk fabric was used as feather surfaces. The model was positioned in the test area using three support wires, which were connected to force balances to measure the lift force acting on it; rods attached to lateral supports at the trailing edge of the model were used to measure drag force. The single flap setup with

flap of 50% chord in length showed significant improvement in post-stall lift recovery as well as a smoother transition on the lift-drag polar graph, though the experiments also showed a reduction in maximum lift coefficient at stall angle.

The drop in maximum lift generation is thought to be due to premature lifting of the flap, causing it to be lifted into the attached flow region. The correlation between the flap incursion into the free stream flow and the drop in lift coefficient can be observed in the polar plot, as the lift data from the flap located further upstream showed a delayed, but more pronounced, drop comparing to the flap located closer to the trailing edge of the wing. Although the lift reduction due to the premature deployment of the passive flap can be alleviated by using porous material, the authors believed that the primary cause of the flap deployment at lower angle of attack is due to the three dimensional flow effect: for an untwisted, rectangular wing, the effective angle of attack at the tip of the wing is lower than that of the center, causing a wedge shaped stall pattern that propagates from the center-line outward; the low aspect ratio of the wing model would lead to a sharper effective angle of attack transition, lifting the flap to a higher position than optimal. The authors also flirted with the idea of dividing the flap in the span-wise direction into small “back-flow pockets” in order to counter the effect of wing tip vortices; the setup resulted in a smoother transition post stall, but also resulted in lower maximum lift coefficient and higher drag due to material fluttering.

The experimental study of the passive flap was expended in by a joint research team consisted of researchers from DLR Berlin, Institute of Bionics

and of Fluid Mechanics at TU-Berlin, and the STEMME Aircraft Company to a higher Reynolds number of 1.25×10^6 , with the results published in Bechert et al. [12] in 1998. The two-dimensional study placed the HQ-17 airfoil model between a two-strut external force balance and used a series of flap material ranging from elastic plastic material to thin metal sheet. During the experiments, it was discovered that the premature deployment of the flap previously documented by Patone and Müller [11] is caused by the increase in static pressure towards the trailing edge of airfoil; the slightly lifted flap would then create a small separation regime behind the flap, causing an increase in drag. The authors suggested three different approaches to correct the protruding flap at low angle of attack: by locking the flap onto the airfoil before flow separation, by using permeable material for the flap, and by introducing jagged trailing edge; the latter two solutions are already present on a bird's wing.

Regarding the positioning of the passive flap along the upper surface of the wing, Bechert et al. [12] discovered that small perturbations on the upper surface of the airfoil at high Reynolds number would result in immediate flow separation if the perturbation is located in the first $60 \sim 70\%$ of the airfoil, thus limiting the installation of the flap hinge to the aft portion of the wing, where minor changes in surface quality do not produce a detectable increase in drag. The authors concluded that the trailing edge of the passive flap should be positioned slightly upstream ($< 1\%$ chord) from the trailing edge of the wing in order to achieve the best result, while the leading edge of the passive flap should not be located too far upstream as the device is only activated when the separation regime propagated to that point. Additional

experiments involving more than one passive flap installed serially in the chord-wise direction were also conducted: under a two-flaps configuration, by designing the upstream flap using porous, flexible material, a design that introduce a fluttering motion, resulted in an increase in lift generation of about 6%. Ultimately, a single passive flap in the aft regime of the airfoil was chosen for full-size aircraft testing as it “does not require any vibration in order to work properly and they are remarkably stable and reliable in their operation. [12]” In order to maintain controllability of the aircraft under a total stall, vortex generators were fitted upstream of the ailerons and elevators, with the passive flap covering only 61% of the wing span. Figure 2.2 shows the passive flap setup used in the first successful application of the passive flap on a full-size aircraft.

Figure 2.2: Schematic of passive flap and vortex generator configuration on the STEMME S10 glider from the Bechert et al. [12] study.

The idea of using passive flap type lift enhancing device was picked up by Bramessfeld and Maughmer [13] at Pennsylvania State University in 2002, where wind tunnel study was performed on a S824 airfoil at $Re = 1.0 \times 10^6$. The experiments were conducted with flaps constructed of 0.35mm thick Mylar tape with flap length of 9% chord; the passive flaps were located at 86% and 70 & 86% chord from the leading edge for single and double spacing flap setups; the aerodynamics constants were measured using pressure tap installed on the airfoil model, and recorded 20% increase in lift. Using the pressure distribution graphs, the authors were able to show that a deployed flap acts as “pressure dams,” allowing lower pressure zone to exist further upstream than on a clean airfoil; the pressure would recover in a stepwise manner as the flow moves downstream, and result in a trailing edge pressure recovery higher than the clean airfoil.

Studies on the passive flap continued at TU Berlin following the Bechert et al. study [12] with computational fluid dynamics (CFD) simulations, and the results were published in 2004 [9]. The simulations were conducted using the TU Berlin in house ELAN code, which is based on implicit second order numerical scheme using the SIMPLE algorithm. Both large eddy simulation (LES) and unsteady Reynolds averaged Navier-Stokes (URANS) simulation were conducted using Rung’s LLR $k-\omega$ turbulence model, as it exhibited the best transitional flow behavior.

As the motion of the flap would change the flow dynamics around the airfoil, two different methods were used to solve for the equilibrium deflection angle of the passive flap: the first is to model airfoil with passive flap of fixed deflection angles, conduct the simulations, and find the zero-moment flap

angle; the second is to combine moment balancing equation with dynamic mesh solver, and simulated the fluid-induced motion. Lift coefficient and flow separation position predictions from both simulation approaches showed good agreement with the experimental data on *HQ17* airfoil model, making numerical simulation the ideal tool for analyzing the flow dynamic within the recirculation flow region. The simulation results showed that the back flow intensity in the recirculation zone is greatly reduced over a wide range of angle of attack, as the flap divides the separation flow region in two, resulting in reduction of overall reverse mass flow. The paper also reported that due to the interaction between cross-flow and recirculation flow, the passive flap is not suitable for adaptation for use on swept wings.

The application of the passive flap on UAV was proposed by Kernstine et al. [14] in 2008, who performed wind tunnel study in the $1 \sim 5 \times 10^5$ Reynolds number range. The two-dimensional experiment was conducted in $14'' \times 10''$ wind tunnel on a NACA2412 airfoil model, with flap material consist of $1/16''$ rigid aluminum plate; the use of either porous material or jagged trailing edge suggested in previous studies [12, 9] was not adapted for this study, possibly due to the thinner material used, and the effect of premature flap deployment can be seen in Figure 2.3 for the lift data with flap closer to the trailing edge.

The initial characterization experiment also includes the testing of more flexible flap material, such as paper, $1/60''$ aluminum foil, and this vinyl plastic. The authors reached the conclusion that rigid flap that cannot conform to the airfoil curvature is not suitable as passive flap, and same for material thick enough to disturb the flow over the airfoil. However,

Figure 2.3: Experimental lift result of a flap with length of $c_f = 0.3c$ from Kernsine et al. [14], the large drop in lift with more upstream flap position (smaller x_f/c) is due to the rigid flap material unable to conform to the curvature of the airfoil.

contrary to previous studies, the authors reported better lift enhancement by attaching strong, flexible plastic flap that conforms to the airfoil curvature closer to the leading edge of the airfoil, although the same configuration also resulted in a worse lift to drag ratio throughout the attached flow range.

Also covered in the study was the application of passive flaps in pair, which cause violent oscillation of the wing at high angle of attack. The application of feathered flap¹ was reported to produce 15% increase in lift with $x_f = 10\%c$, but with significant increase in drag, though not as much as the single flap configuration. The study between single flap configurations at $Re = 4.53 \times 10^5$ and $Re = 1.66 \times 10^5$ showed that the optimal passive flap configuration is Reynolds number sensitive.

¹Passive flap split into multiple sections in the span-wise direction.

2.3 Post-2008 Studies

The current project began in mid-2008 with emphasis on the development of the numerical code for simulation of the passive flap for application on MAV, the task of collecting experimental data was allocated to undergraduate student Tan Jing Ru as a part of his final year project; the analyzed watertunnel data was published in 2009 by Professor Schlüter [15]. The experimental study was conducted inside the Nanyang Technological University Watertunnel Lab using solid metal airfoil models manufactured from a solid block of steel or aluminum using CNC wire-cut machine, and rigid carbon-fiber strip as the flap. Studies were performed at $Re = 30,600$ for NACA0012 and NACA2213 airfoil model, and $Re = 40,000$ on SD8020 and NACA4412 airfoils, and showed close to 50% increase in lift with a single $c_f = 0.4c$ flap located at $x_f = 0.6c$; installing two $c_f = 0.2c$ flaps at $x_f = 0.6c$ and $x_f = 0.8c$ yielded the same gain in lift. Figure 2.4 shows the results from the SD8020 experiments.

Results from Figure 2.4 forms the basis of our simulation code development, which uses the lift data to determine the accuracy and validity of simulation results. Due to limitation of the force measurement instruments, however, all drag force data collected fell within the margin of error of the instrument ($\sim 10.05N$), and could not be used for drag polar and lift-drag ratio analysis. Regardless, this study formed the first step in our investigation into the application of passive-flap high lift device on MAV.

In 2011, Johnston et al. [16] presented oil flow visualization with passive flap installed at $Re = 4 \times 10^5$ on a custom Natural-Laminar-Flow (NLF)

Figure 2.4: Lift curves of various passive flap configurations installed on SD8020 airfoil. [15]

airfoil as well as c_p measurements. The oil flow visualization with fixed passive flap deployment angle (β) indicated that flow separation on an airfoil with passive flap occurs further downstream comparing to the clean airfoil; the oil flow pattern also showed that separated flow would reattach itself onto the upper flap surface, with reverse flow region between the separation point and the leading edge of the flap.

Chapter 3

Experimental Setup

Experimental studies were performed in two phases: the first phase is to recreate the results published by Schlüter [15] using the NTU watertunnel system, while gathering information that could help in the development of the CFD code; the second phase is to evaluate the effect of three dimensional flow effect on the passive flap using the Low Reynolds Number Wind tunnel facility. The following sections contain information regarding model manufacturing, instrument setup, software interface, and specification of the facility used in the wind tunnel experiments conducted in Summer 2011.

3.1 Overview of Wind Tunnel Facility

The low Reynolds number wind tunnel system available for the experiment consists of the Wind tunnel manufactured by Long-Win Science and Technology Corporation in Taiwan and the two-axis model positioning system manufactured by Chroma International. The wind tunnel system is housed

in the NTU aircraft hanger, with close proximity to the material lab and graduate student offices; due to the possible structural damage to the hanger door as well as noise control problem, the maximum test-section speed of the windtunnel is limited to $12m/s$. Table 3.1 listed the specification of the wind tunnel, the model positioning system as well as the integrated 6-component force balance.

Table 3.1: Specification of the NTU Wind Tunnel System

LW-3890 Wind Tunnel	
Test Section	$1.1(W) \times 0.9(H) \times 2.0(L)m$
Speed Range	$3-30 \text{ }^m/s$
Flow Quality	• Turbulence Intensity < 1
	• Flow Angularity $< \pm 1^\circ$
	• Uniformity $> 98.5\%$
MPS-750-2A 2 Axis Positioning System	
Model Fixture Height	$0.4m$
Pitch Angle Range	$-25^\circ \sim 25^\circ$
Pitch Resolution	0.1°
Yaw Angle Range	$-35^\circ \sim 35^\circ$
Yaw Resolution	0.1°
IB6C-10 Internal Force Balance	
Maximum Normal Force	$50 \text{ } N$
Maximum Side Force	$50 \text{ } N$
Maximum Axial Force	$50 \text{ } N$
Maximum Pitching Moment	$5 \text{ } Nm$
Maximum Yawing Moment	$5 \text{ } Nm$
Maximum Rolling Moment	$5 \text{ } Nm$
Resolution	0.1% of Full Scale

The flow speed inside the test section is controlled by adjusting the fan motor frequency, and monitoring the resulting airspeed reading from a Pitot static probe. All readings from the Pitot probe, pitch and yaw controller, and 6-components internal force balance are feed into a LabView based

data acquisition and control platform. The raw data and aerodynamics coefficients can be exported in comma-separated-value (.csv) file format; compilation of lift curves and curve fits are post-processed using OpenOffice¹ Calc. The force balance was re-calibrated before the experiment began, and zeroed before each set of tests. Figure 3.1 below shows the wind tunnel and the model controller

(a) Wind Tunnel and Control Station

(b) Close-up of Internal Force Balance

Figure 3.1: (a) Wind tunnel with model control and force measurement system, and (b) close up of the strain gauges on the 6-components internal force balance.

¹Converted to LibreOffice after January 2011

3.2 Rectangular Wing Model

3.2.1 Manufacturing of Wing

SD8020 airfoil profile was chosen for the three-dimensional passive flap experiment due to the availability of prior experimental data [15, 17]. The model was designed to allow for easy swapping of different wing profiles in the future, and consists of two parts: the aluminum mount, and the wing model itself. The wing mount is manufactured from a solid block of aluminum using milling and drilling machine, and contain two main parts: the internal force balance adapter, and a flap platform for the wing model to be fastened on, as shown in Figure 3.2 below.

Figure 3.2: Aluminum mount that connects the wing to the internal force balance of the wind tunnel.

The wing model was manufactured with balsa-wood based spar and rib construction. The wing is $0.1m$ in chord and $0.53m$ in span with a $0.03m$ platform in the center for attachment to the aluminum mount. $0.5mm$ thick balsa planks were used for skin, and a $2mm \times 2mm$ balsa stick sanded down to shape was used for the leading edge. Figure 3.3 shows the upper side of the wing model without the foam fuselage that encased the connector.

Figure 3.3: Balsa wing model used in the three-dimensional wind tunnel experiments. The spar can be seen crossing the connector block.

The wind tunnel blockage correction for the rectangular wing model is estimated to be $\epsilon_t = 0.00126$ for blockage correction based on Equation 3.1, which resulted in 0.13% change in effective flow velocity and well below the sensitivity of the force balance.

$$\epsilon_t = \frac{\Delta V}{V_u} = \frac{1}{4} \frac{\text{model frontal area}}{\text{test-section area}}, \quad (3.1)$$

and the downwash correction was estimated to be $\delta \sim +0.125$, which would result in less than 0.4° increase in effective angle of attack throughout. The estimations were based on equations and plots from Low Speed Wind Tunnel Testing by Barlow et al. [18]. Overall, the enclosed test section has minimal effect on the flow field around the wing model and the force measurements.

The balsa construction method was chosen over producing the model with CNC foam cutter due to issue with model accuracy with the foam cutter: the heat from the foam cutter would often cause deformation of the

wing close to the trailing edge. The accuracy of the instrument as well as the wing model can be demonstrated by comparing the wind tunnel results with existing data. Since the available low Reynolds number data for SD8020 were recorded in two-dimensional coefficients [17], finite-wing correction based on lifting line theory (Equation 3.2) must be applied to make it comparable with our data.

$$C_{L,\alpha} = \frac{C_{l,\alpha}}{57.3 \times \frac{C_{l,\alpha}}{\pi e AR}} \quad (3.2)$$

The efficiency factor $e = 0.9$ was used to correlate the two data sets, with the results shown in Figure 3.4. Note that the lifting-line theory is only accurate for the pre-stall (linear) portion of the lift curve.

Figure 3.4: Resulting lift curve from the SD8020 wing model compared with the lift curve from Selig et al. [17] with finite-wing correction.

3.2.2 Passive Flap Setup

Rigid carbon fiber composite strip of $0.3mm$ in thickness with a density of $1.27 \times 10^3 kg/m^3$ was used in the wind tunnel experiments. The material was chosen for its directional strength and the availability of the material from previous experiments [15]. The grain of the carbon fiber strip was aligned in the span-wise direction to avoid shearing due to non-uniform wing loading, while making it easier to cut down to the desired flap length (c_f) without fracturing. Figure 3.5 shows the carbon fiber strip used for flap.

Figure 3.5: Carbon fiber material used for flap. Each grid is $0.5cm$.

The thickness of the flap has been shown to have no significant effect on the aerodynamics property of the airfoil at $Re = 40,000$ with the flap taped down onto the wing surface and while the wing is under attached flow condition (prior to the flap deployment) [15]. The passive flaps were installed onto the wing model using strips of transparent office tape applied at the mid-point and either ends of the flap. This arrangement was made due to the elasticity of the tape forcing the flap down, delaying or preventing the deployment of the flap. The now porous “hinge” region has the added benefit of encouraging pressure equalization across the flap in the attached

flow regime and preventing premature deployment of the flap.

The flap configurations were setup to test the influence of three dimensional flow effect, especially the wing-tip vortex, on the performance of the passive flap. The flaps were installed in sets of two with each flap spanning out from the root of the wing at different flap spans (b_f) with the expectation that flap with a shorter span would be under less influence from the downwash comparing to longer flaps; reducing the span by too much, though, would render the flap ineffective. Another parameters tested in this experiment is the flap length (c_f): increasing c_f is believed to help divide the recirculation region, reduce the overall size of the separation bubble, and generates more lift. However, the advantage of increasing c_f will eventually be neglected by the higher inertia of the flap [12]. The role of x_f on flap performance is less clear in the literature, and while earlier experiments suggested that it is more beneficial to flush the flap with the trailing edge of the wing [12], later studies showed that better performance can be achieved by placing the flap close to $x_f = 0.5c$ [14, 15].

Experiments were performed with flap lengths of $c_f = 0.1c$, $0.2c$, and $0.3c$, starting from the trailing-edge flush position² and move up to $x_f = 0.5c$ in $0.1c$ increments. The experiments were repeated for $b_f = 1.0b_w$, $0.8b_w$, $0.6b_w$, $0.4b_w$, and a composite configuration with $b_f = 0.6b_w$ flap flushed to the root of the wing and $b_f = 0.4b_w$ on the outer portion. Before each experiments, data were taken with the flap taped down to determine the effect of the flap thickness on clean airfoil. Figure 3.6 is a visualization of the different flap spans, lengths, and positions used in the experiment.

²e.g. $x_f = 0.9c$ for $c_f = 0.1c$ the flap.

Figure 3.6: Flap Configurations Tested on the Rectangular Wing

3.3 Adaption on a Remote-Control Aircraft

The setup of the wing model experiment was designed to assist us in understanding of the behavior and performance of the passive flap device at low Reynolds number. However, the ultimate goal of the study was to study the feasibility of applying the passive flap on an existing MAV platform. To that end, we have acquired a remote-control airplane as our test platform.

3.3.1 Aircraft Model

A remote control (RC) aircraft model of foam construction was chosen due to the limited reading range of our internal force balance as well as availability of the model in the lab from a previous project. The model used in the practicality study is the Merlin electric racer/glider made by Multiplex, which has a wing span of 0.783 m and a total surface area of 0.113 m^2 [19]³. Due to the flexibility of its foam construction, the wing would deform and curve upward as angle of attack increases and inhibits the movement of the passive flap, and severe wing fluttering was observed as the aircraft approaches stall. The structural problems were resolved by the installation of additional carbon fiber spars. The tail section of the aircraft model was removed to install the connector for the internal force balance, which is positioned at the new center of mass. Figure 3.7 shows the model mounted in the wind tunnel ready for testing.

³Which most likely include the surface area of the horizontal stabilizer, as the actual wing area measured is $7.49 \times 10^{-2}\text{m}^2$.

Figure 3.7: Picture of the remote control aircraft model inside the wind tunnel mounted through the cut-off tail section.

The larger RC model required a larger blockage correction of $\epsilon_t = 0.0042$, i.e. the effective free stream velocity is 0.42% larger than the measured velocity; the boundary correction factor for the model in closed rectangular test section is $\delta = 0.15$, which translate to $\sim 0.6^\circ$ increase in effective angle of attack.

3.3.2 Passive Flap Configurations

Based on the results from the rectangular wing experiments, we have chosen the flap length $c_f = 0.3c$ for the RC model experiments. The span of the flap is constrained by the length of the structural spar, which limits the deformation of the wing, to a $b_f = 0.18m$; the curved wing at high angle of attack would cause the flap to lock up as the material would bulge causing the flap hinge to become misaligned. Tests were conducted for flap positions of $x_f = 0.7c$ and $x_f = 0.6c$.

Due to the larger chord length of the Merlin, the carbon fiber material from the previous experiments could not be used in these tests. Instead, the flaps were cut out from a $0.25mm$ thick bi-directional carbon fiber sheet, which has a density of $1.17 \times 10^3 kg/m^3$. The flaps were secured to the model in similar fashion as from the previous experiments with the rectangular wing. The material used for the passive flap is shown in Figure 3.8.

Figure 3.8: Flap material used for the RC model experiments. Each grid is $0.5cm$.

Chapter 4

Flow Solvers and Numerical Methods

4.1 Simulation Strategies

The simulation of the passive device is complicated by the fact that the positioning of the flap depends on the recirculating flow in the separation zone, while the size and strength of the recirculation flow field depends on the upper-airfoil geometry. Simulations conducted by Schatz et al. [9] inspired the adaptation of the open source OpenFOAM flow solver¹ used for the current research via two different approaches: the first approach is to perform steady-state simulation over a range of flap angles for the passive flap; the second approach is to conduct unsteady simulation using a limited-fluid structure interaction solver.

¹The core OpenFOAM flow solver was used to obtain the flow solution while additional numerical algorithms were developed to model the movement and deployment of the flap.

4.1.1 Steady-State Simulation Based Method

Thanks to the built-in automatic mesh generator, OpenFOAM solvers can be modified to conducting optimal flap angle search based on results from its steady-state turbulence solver. Instead of evaluating flow fields over a range of fixed flap angles as in previous study [9], the new solver resolves the equilibrium flap angle by performing iterative optimization with the flow results from two initial flap angles. New mesh is generated for the each of the new flap angles, and the new simulation result is feed back into the optimization subroutine. The simulation ends when the moment acting on the flap falls within the user defined tolerance from zero, indicating that the steady-state flap deflection angle (β) has been reached. Figure 4.1 below shows the simulation strategy used in the development of the code.

Figure 4.1: Strategy for steady-state simulation of the passive flap.

4.1.2 Transient Simulation Based Method

The concept behind the transient simulation is very straightforward comparing to the steady-state approach: the flap is moved based on moment on the flap and governing equation of motion after every time step, and the old mesh adapt to the new flap position using the built-in dynamic-mesh updating algorithm of OpenFOAM. The transient solution also has the ability to run the simulation using the more sophisticated LES turbulence model, as well as better ability to monitor the effect on lift from unsteady fluttering of the passive flap at high angle of attack that was observed in the experiments. The disadvantage of the transient simulation comparing to the steady-state one is the need to develop new fluid-induced motion (FIM) solver based on OpenFOAM libraries instead of simply combining existing ones, resulting in significantly longer development time.

4.2 Overview of OpenFOAM

The solvers used in the current project are developed based on the open-source C++ continuum mechanics solver Open Field-Operation And Manipulation (OpenFOAM) software suite. The core solver has been validated[20, 21], and used in fluid dynamics research[22, 23] under various flow conditions; the solver was also adapted by Volkswagen Group as AeroFOAM for research and development for their VW and Audi brands[24]. As an open-source project the OpenFOAM source code is continuously evolving; in order to maintain the compatibility of simulation results, and to prevent feature creep, all steady-state simulations were conducted using OpenFOAM

version 1.6.x while all transient simulations were conducted with modified solver using the libraries from version 1.7.x. The following sections provide an overview of the numerical methods used in OpenFOAM as well as the development of the solvers used for the current study.

4.2.1 Development of OpenFOAM

The OpenFOAM libraries were developed in C++ to take advantage of the object oriented programming and operator overload features not available in older programming languages such as C and FORTRAN [1]. The object oriented programming (OOP) allows the developers to encapsulate relevant information of a flow property into a single class object, such as the scalar values and unit dimensions of flow velocity U , and the operator overload feature enabled the developers to implement tensor algebra in a manner that resembles mathematical notations. As the unit dimension is included within the object that defines each variable, the solver will automatically check for unit consistency in the case where user defined variable or equations were introduced.

Due to the way the variables are setup in OpenFOAM, two additional tensor-derivative classes are implemented in OpenFOAM: finite volume calculus (fvc class operators) which performs tensor differential operations and returns a tensor field, and finite volume method (fvm class operators), which produce a matrix representative of the tensor operation that form part of the equation that can be solved to advance the simulation by a time step; both operation classes contain separated operators for $\partial U/\partial t$, $\nabla \cdot$, ∇ , $\nabla \times$,

and ∇^2 , which allow for better discretization practice for the operators. An example of implementation of equations using the new tensor class structure is shown below with the transcription of the conservation equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\phi) = 0, \quad (4.1)$$

where $\phi = \rho U$, into OpenFOAM format:

```
fvMatrixScalar rhoEq
(
  fvm::ddt(rho)+fvc::div(phi)
);
...
rhoEq.solve();
```

The matrix “rhoEq” contains information for all three components from the left hand side of the conservation equation, and the unknown variable is solved using the “rhoEq.solve()” command that seeks the value for the matrix to equal to zero.

4.2.2 Implementation of Navier-Stokes [1]

The object oriented approach employed by OpenFOAM greatly simplified the process of implementing the Navier-Stokes equation into the continuum solver, as the implementation of the equation can now be accomplished separated from the discretization process. The incompressible Navier-Stokes equation,

$$\frac{\partial U}{\partial t} + \nabla \cdot (U \otimes U) - \nabla \cdot (2\mu \nabla U) = -\frac{1}{\rho} \nabla p, \quad (4.2)$$

is implemented as

```
fvMatrixVector Ueqn
(
    fvm::ddt(U)
    +fvm::div(phi, U)
    -fvm::laplacian(nu, U)
);
...
solve(Ueqn == -fvc::grad(p));;
```

By combining the Navier-Stokes equation with the conservation equation (Equation 4.1), we are able to obtain a system of equations that solves for flow velocity and pressure². As the momentum equation contains the non-linear convection term $\nabla \cdot (U \otimes U)$, the solution can only be obtained by either using the more resource intensive non-linear solver or by linearizing the convection term as in Equation 4.3:

$$\begin{aligned}
 \nabla \cdot (U \otimes U) &= \sum_f S \cdot (U)_f (U)_f \\
 &= \sum_f F(U)_f \\
 &= a_P U_P + \sum_N a_N U_N,
 \end{aligned} \tag{4.3}$$

where the flux F , a_P , and a_N are functions of U , S is the outward-pointing face area vector, $(U)_f$ is the velocity flowing through that face, and U_P is the flow velocity at an arbitrary point P. Using Equation 4.3, the momentum equation can be rewritten as

²As the fluid density is constant in incompressible flow, the kinematic pressure $p_k = p/\rho_0$ is used in the in place of pressure. The actual pressure and force can be recovered in the post-processing process.

$$a_P U_P = H(U) - \nabla p, \quad (4.4)$$

with

$$H(U) = - \sum_N a_N U_N + \frac{U^0}{\Delta t}, \quad (4.5)$$

where U^0 is the velocity from the previous time step.

The semi-discretized Navier-Stokes equation can then be rearranged and combined with the discretized continuity equation to form the pressure equation

$$\begin{aligned} \nabla \cdot \left(\frac{1}{a_P} \nabla p \right) &= \nabla \cdot \left(\frac{H(U)}{a_P} \right) \\ &= \sum_f S \cdot \left(\frac{H(U)}{a_P} \right)_f, \end{aligned} \quad (4.6)$$

the satisfaction of which would ensure mass conservation.

The combination of Equation 4.6 and Equation 4.4 will give us the fully discretized incompressible Navier-Stokes momentum and conservation equation in the following form:

$$a_P U_P = H(U) - \sum_f S(p)_f, \quad (4.7)$$

$$\sum_f S \cdot \left[\left(\frac{1}{a_P} \right)_f (\nabla p)_f \right] = \sum_f S \cdot \left(\frac{H(U)}{a_P} \right)_f. \quad (4.8)$$

While the discretization process eliminate the non-linearity of the convection term from the solution process, it introduced pressure-velocity coupling into the equations. Direct solution of the the coupled variables would require solving the matrix containing Equation 4.8 and Equation 4.7, which is computationally expensive for large number of grid points. A less expensive way of solving for the coupled variables is to separate the solution process: the velocity field is solved first using guessed values for pressure field, which is then used to solve for the pressure equation to ensure conservation of mass. The calculation process can be further simplified by substituting the two variable field with pressure and velocity correction terms (p' and u' in Equation 4.9),

$$u_i^m = u_i^{m*} + u' \text{ and } p^m = p^{m-1} + p', \quad (4.9)$$

where p^{m-1} is the initial guess value for the pressure field from the previous time step ($m - 1$) used for the estimation of the intermediate velocity field u^{m*} . By substituting Equation 4.9 into Equation 4.8 and Equation 4.7, we obtain the pressure-correction equation,

$$\frac{\delta}{\delta x_i} \left[\frac{\rho}{A_P^{u_i}} \left(\frac{\delta p'}{\delta x_i} \right) \right]_P = \left[\frac{\delta(\rho u_i^{m*})}{\delta x_i} \right]_P + \left[\frac{\delta(\rho \tilde{u}'_i)}{\delta x_i} \right]_P, \quad (4.10)$$

where $\tilde{u}'_i$ is the velocity correction term from neighboring cells. The treatment of the above mentioned term is the main difference between the iterative algorithms used by OpenFOAM for steady-state and transient problems, which will be explained in the following sections.

SIMPLE Algorithm

When solving for a steady-state problem, the emphasis is placed on the speed of reaching the steady-state solution instead of the accuracy of the history term in the intermediate steps, neglecting the need to fully resolve the pressure-velocity coupling and allows for larger time step sizes. This means that changes between consecutive solutions are no longer small, and the non-linearity of the system becomes more important. The steady-state solvers in OpenFOAM are constructed around the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm [25], which simplified the solution by neglecting the $\tilde{u}'_i$ term, an unknown variable, from Equation 4.10. This move was justified as the omission does not affect the correctness of the converged solution, as the final pressure field would produce a velocity field that satisfied the continuity equation [25]. The solution procedure takes the following steps:

1. Set the boundary conditions.
2. Solve for the intermediate velocity field with guessed initial pressure.
3. Compute the mass flux through the cell faces.
4. Solve the pressure equation and apply under-relaxation.
5. Correct the mass flux at cell faces
6. Repeat step 4 and 5 for prescribed iterations of non-orthogonality correction.
7. Correct the velocity based on the new pressure field.
8. Update the boundary condition.
9. Repeat the iterative process until convergence is achieved.

PISO-SIMPLE Hybrid Algorithm

Comparing to the steady-state flow problems, the transient (unsteady) flow problem requires a higher level of accuracy for each time step. This can

be accomplished either by employing an algorithm that is more accurate per time step or by looping the algorithm in sub-steps until convergence is reached for each of the time step. The former can be achieved by employing the Pressure Implicit with Splitting of Operators (PISO) algorithm which was developed as a non-iterative way of handling the velocity-pressure coupling by Issa [26]. Like the SIMPLE algorithm, PISO neglect the $\tilde{u}'_i$ term in Equation 4.10 in order to acquire the velocity and pressure corrector; unlike the SIMPLE algorithm, the corrected fields are used to construct a secondary corrector, using the $\tilde{u}'_i$ term from the first correction field. Compared to SIMPLE-based transient solvers, PISO algorithm requires more computing power per iteration, but is faster per time-step as only a single iteration is involved. In practice, though, PISO algorithm requires very small time step size to maintain the stability of the simulation.

The OpenFOAM solver that forms the backbone of our fluid-induced motion solver was constructed based on a hybrid PISO-SIMPLE approach (PIMPLE), combining the accuracy aspect of the PISO algorithm and the iterative approach from the SIMPLE algorithm that allows for larger time-step size and under-relaxation between the iterations; the result is an algorithm that maintains stability at larger time step and Courant number, while requiring less sub-step iterations to achieve convergence. The solution procedure used in the OpenFOAM implementation of PIMPLE algorithm takes the following form ³:

1. Set the boundary conditions.

³Steps 1-9 from PIMPLE algorithm is identical to the procedure for PISO algorithm. The algorithm can be set to run in PISO mode by setting the number of outer-corrector loop to one.

2. Solve for the intermediate velocity field with guessed initial pressure.
3. Compute the mass flux through the cell faces.
4. Solve the pressure equation.
5. Correct the mass flux at cell faces
6. Repeat step 4 and 5 for prescribed iterations of non-orthogonality correction.
7. Correct the velocity based on the new pressure field.
8. Update the boundary condition.
9. Repeat from step 3 for prescribed number of iterations (inner correction loop).
10. Repeat from step 1 with under-relaxation applied to the pressure field for prescribed number of iterations (outer correction loop) or until convergence is reached.
11. Increase the time step and repeat the process.

4.2.3 Turbulence Modeling

OpenFOAM solvers were developed to employ both the Reynolds Average Navier-Stokes equation (RANS) and Large Eddy Simulation (LES) simulations within the same solver. This is achieved by the implementing the Navier-Stokes equation in the following form:

```
fvMatrixVector Ueqn
(
    fvm::ddt(U)
    +fvm::div(phi, U)
    +turbulence->divDevReff(U)
);
...
solve(Ueqn == -fvc::grad(p));;
```

where the “turbulence→divDevReff(U)” pointer forwards the velocity field to user selected turbulence model, which returns the Reynolds stress tensor by evaluating the effective viscosity $\nu_{eff} = \tilde{\nu} + \nu_t$. The turbulence models that were used in the present study are listed in the following sections.

Spalart-Allmaras RANS Model

The Spalart-Allmaras turbulence model is an one equation linear eddy viscosity model for RANS simulations, which solves for a single equation for the viscosity-like variable $\tilde{\nu}$. The implementation of the Spalart-Allmaras model in OpenFOAM is based on the original equations from 1994 [27] and incorporates the modifications proposed by Ashford [28] in order to maintain simulation stability for flow around complex geometries. The resulting formulation is known as the SA-fv3 model, though the OpenFOAM implementation has had the tripping terms removed. The calculation of the turbulence viscosity $\tilde{\nu}$ is based on the following equation:

$$\frac{D\tilde{\nu}}{Dt} = c_{b1}\tilde{S}\tilde{\nu} + \frac{1}{\sigma} \left[\nabla \cdot ((\nu + \tilde{\nu})\nabla\tilde{\nu}) + c_{b2}(\nabla\tilde{\nu})^2 \right] - c_{w1}f_w \left[\frac{\tilde{\nu}}{d} \right]^2. \quad (4.11)$$

The solution of the above equation can be related back to the momentum equation via

$$\mu_t = \rho\tilde{\nu}f_{v1}, \quad (4.12)$$

where the damping function

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}, \quad (4.13)$$

$$\chi = \frac{\tilde{\nu}}{\nu}, \quad (4.14)$$

The production term from Equation 4.11 is defined as

$$\tilde{S} = f_{v3}\Omega + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}, \quad (4.15)$$

where Ω is the magnitude of vorticity, and

$$f_{v2} = \frac{1}{(1 + \chi c_{v2})^3}, \quad (4.16)$$

$$f_{v3} = \frac{(1 + \chi f_{v1})(1 - f_{v2})}{\chi}; \quad (4.17)$$

the near-wall inviscid destruction term is defined as

$$f_w = g \left[\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{\frac{1}{6}}, \quad (4.18)$$

where

$$g = r + c_{w2}(r^6 - r), \quad (4.19)$$

and the non-dimensional mixing length is

$$r = \min \left[\frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2}, 10 \right], \quad (4.20)$$

with an artificial limiter on $\tilde{S}$ to ensure that it remains positive. With the value of $\tilde{\nu}$ known, we can construct the momentum equation source term `divDevReff()` by combining the laminar viscosity ν and $\tilde{\nu}$. The values of constants used for the turbulence model in our study are listed in Table 4.1 below.

The current study employs the Spalart-Allmaras model when testing the

Table 4.1: Constants for Spalart-Allmaras Turbulence Model

Constants	Value
c_{b1}	0.1355
c_{b2}	0.622
c_{w2}	0.3
c_{w3}	2.0
c_{v1}	7.1
c_{v2}	5.0
σ	0.66666
κ	0.41

FIM code as it is less resource intensive comparing to the SST model.

k - ω SST RANS Model

The k - ω Shear-Stress Transport (SST) model is a two equation turbulence model developed by Menter [29, 30] that models the turbulence property by incorporating two additional transport equation that solves for k , the turbulence kinetic energy and ω , the specific dissipation per unit of k . The SST model addresses the difficulty with the prediction of μ_t under free stream condition by earlier k - ω models by incorporating the standard k - ϵ model using a blending function. The version of the incompressible k - ω SST model⁴ implemented in OpenFOAM follows the formulation described in Menter & Esch⁵ [31], and included the blended ω wall treatment for $y^+ > 1$. The transport equations used in OpenFOAM are

⁴All terms in the transport equations are divided by the constant ρ .

⁵The omega production term used in this version is expressed as αS^2 , while the more recent SST models use $\alpha \tilde{P}_k / \nu_t$ instead.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tilde{P}_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_t) \frac{\partial k}{\partial x_j} \right], \quad (4.21)$$

and

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, \quad (4.22)$$

where the production limiter is defined as

$$\tilde{P}_k = \min(P_k, c_1 \beta^* k \omega); P_k = \nu_t \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right); \quad (4.23)$$

the k - ϵ / k - ω blending function F_1 is defined as

$$F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho \sigma_{\omega 2} k}{CD_{k\omega} y^2} \right] \right\}^4 \right\}, \quad (4.24)$$

with

$$CD_{k\omega} = \max \left(2\rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right); \quad (4.25)$$

the turbulent eddy viscosity is defined as

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)}, \quad (4.26)$$

where the blending function

$$F_2 = \tanh \left[\left[\max \left(\frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right) \right]^2 \right], \quad (4.27)$$

and the strain rate invariant $S = \sqrt{2S_{ij}S_{ij}}$. All constants used in the transport equation, except for a_1 , β^* , and c_1 , are computed in a blended form that combines the k - ϵ and k - ω constants via $\phi = F_1\phi_1 + (1 - F_1)\phi_2$. The value of constants used are listed below in Table 4.2.

Table 4.2: Constants for k - ω SST Turbulence Model

Constants	Value	Constants	Value
a_1	0.31	β_2	0.0828
β^*	0.09	σ_{k1}	0.85034
c_1	10.0	σ_{k2}	1.0
α_1	0.5532	$\sigma_{\omega1}$	0.5
α_2	0.4403	$\sigma_{\omega2}$	0.85616
β_1	0.075		

The SST model was chosen for our simulation for its robustness and performance at both $y^+ > 30$ and $y^+ < 1$. As k - ω models do not require wall damping at boundary layer, standard wall function is used for the coarse grids of $y^+ \sim 30$ and no wall function is used for the $y^+ < 1$ simulations. Additionally, steady-state simulations conducted with NACA2231 airfoil and $c_f = 0.2$, $x_f = 0.8$ flap configuration showed that k - ω SST offers better prediction for passive flap simulation comparing to Spalart-Allmaras and k - ϵ models, as shown in Figure 4.2.

As shown in Figure 4.2, both k - ϵ and Spalart-Allmaras predicted a negative contribution to lift by the passive flap at $\alpha = 10$ comparing to airfoil with no flap attached; additionally, both k - ϵ and Spalart-Allmaras models produced highly irregular lift prediction between $\alpha = 14$ and $\alpha = 20$.

Figure 4.2: Comparison of turbulence models for prediction of lift by NACA2231 airfoil with flap.

Dynamic Smagorinsky LES Model

Unlike RANS turbulence models, which calculate the turbulence source term using the Reynolds stress term modeled with turbulence viscosity, the Large Eddy Simulation (LES) reduces computational load of solving the full Navier-Stokes equations by filtering out the sub-grid turbulence. One of the more popular LES that utilizes the Sub-Grid Scale (SGS) model for the filtered terms is the Smagorinsky model [32], which assumes that the small scale eddies are in equilibrium. The Smagorinsky model expresses the sub-grid turbulence viscosity as $\nu_{SGS} = (C_S \Delta)^2 |\bar{S}|$, where $|\bar{S}| = \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$ is the magnitude of the large-scale strain-rate tensor, Δ is the LES filter width, and C_S is the Smagorinsky Constant; similar treatment can be used to calculate the residual kinetic energy $k = (C_I \Delta)^2 |\bar{S}|^2$, where C_I is also a constant. The two values and the laminar viscosity combined to form the

effective turbulence stress tensor via the Boussineq approximation, which is then used to calculate the source term of the momentum equation. One drawback of the Smagorinsky model is that there is no universal C_S value that could be used for all flow conditions. Germano et al. [33] proposed a dynamic method of generating the constants C_{dynS} and C_{dynI} that account for both spatial and temporal differences in flow conditions using locally averaged flow properties. The implementation of the Dynamics Smagorinsky Model in OpenFOAM incorporates the error minimization steps by Lilly [34] and solves for the C_{dynS} constant using

$$C_{dynS}^2 = \frac{L_{ij}M_{ij}}{M_{ij}^2}, \quad (4.28)$$

where the dynamic resolved turbulent stress is

$$L_{ij} = \tilde{u}_i \tilde{u}_j - \widetilde{u_i u_j}, \quad (4.29)$$

and the dynamic sub-grid scale stress model is

$$M_{ij} = -2\Delta^2(\alpha^2|\tilde{S}|\tilde{S}_{ij} - |\tilde{S}|\widetilde{S_{ij}}), \quad (4.30)$$

where $\alpha = 2$; similarly, the constant C_{dynI} can be obtained by

$$C_{dynI}^2 = \frac{K_{ij}m_{ij}}{m_{ij}^2}, \quad (4.31)$$

where the dynamic turbulence kinetic energy is

$$K_{ij} = 0.5(\overline{|U|^2} - |\bar{U}|^2) \quad (4.32)$$

and the dynamic eddy viscosity sub-grid stress is

$$m_{ij} = 0.5\Delta^2(\alpha^2|\bar{S}|^2 - \overline{|S|^2}). \quad (4.33)$$

We have chosen to use the Dynamic Smagorinsky model for this study due to the faster computational time and the accuracy of the model when applied to low Reynolds number separated flow [35, 36].

4.2.4 Handling of Flap Motion

In order to implement the simulation strategies outlined in section 4.1, we must first find a way to handle the motion of passive flap. In the case of the steady-state solver, the equilibrium flap angle is obtained by running optimization algorithm that seeks the flap position that yields zero moment; this requires flow solutions of multiple fixed-flap angle simulations, and requires the utilization of the script based mesh generator utility `snappyHexMesh` to automate the process. On the other hand, the equilibrium flap angle could also be obtained by introducing fluid-induced motion and combining the transient flow solver and automatic mesh motion solver from OpenFOAM. The following sections provide details on the mesh generating utility and the automatic mesh motion solver.

snappyHexMesh Utility

It should be noted that the details provided on the automatic mesh generation utility **snappyHexMesh** in the following section is based on the information presented in the OpenFOAM user guide[37] as well as personal experience, and only covers the feature that was utilized in the current study.

The **snappyHexMesh** utility is used in conjunction with the Stereolithography (STL) file that defines the simulation model and the rectangular, uniform sized, mesh generated using the **blockMesh** utility that defines the mesh domain; the rectangular flow domain makes up the background mesh for which the STL model is superimposed upon (see Figure 4.3(a)). The utility reads in the surface defined by the STL file and perform localized refinement in the vicinity of the STL surface based on user defined parameter, with each level of refinement reducing the subsequent cell edges by half (as shown in Figure 4.4(a)); mesh refinement is also performed in user defined “refinement box” as shown in Figure 4.3(b), providing finer mesh resolution in region where detailed flow structure is of interest. The surface based mesh refinement level shown in Figure 4.3 and 4.4 were performed with maximum refinement level of 6 and minimal refinement level of 5, which corresponds to an average y^+ of 25. Following the mesh refinement, cells that fully fall within the space defined by the STL surfaces are removed; the remaining cell vertices closest to the STL surface are then repositioned, or “snapped”, onto the surface, as shown in Figure 4.4(c).

(a) Background Mesh

(b) Localized Mesh Refinement

(c) Mesh Snapping

Figure 4.3: Overall mesh generation process by `snappyHexMesh` starting from background mesh generated by `blockMesh` utility.

(a) Background Mesh

(b) Localized Mesh Refinement

(c) Mesh Snapping

Figure 4.4: Detailed mesh generation process by `snappyHexMesh` starting from background mesh generated by `blockMesh` utility.

Automatic Mesh Motion

The automatic mesh motion algorithm in OpenFOAM plays a substantial role in the development of our unsteady-flow FIM solver, as it links the flap-motion solver to the flow solver. The mesh motion in OpenFOAM is computed via a polyhedral vertex-based approach[38, 39], which solves for the vertex displacement vector d_u using the Laplace equation

$$\nabla \cdot (\gamma \nabla d_u) = 0, \quad (4.34)$$

where γ is the user defined displacement diffusivity; the new position of the mesh vertex is computed at the start of each time step. Due to the file structure of the OpenFOAM simulation directories, the mesh points and connectivity information is stored separately and does not change over time; instead, the changes in position of the mesh points relative to the original mesh are stored as **pointDisplacement** vector field in the output directories. There are several options available in OpenFOAM for the calculation of γ for displacement based Laplace equation: uniform directional, where $\gamma = \text{const.}$; inverse distance based calculation $\gamma = 1/l^n$, where l is the distance to the moving boundary patch and n is a positive exponent; and the exponential based calculation $\gamma = e^{-l}$. The difference between the diffusivity options are illustrated using the grid setup in Figure 4.5(a), which consists of a $1m$ flap hinged $0.25m$ below the upper-aft edge of a $0.5m \times 0.5m$ square; the free stream flow of air coming from the left hand side at a inlet speed of $2.0m/s$ and the asymmetrical placement of the flap should produce a flow field that pushes the flap downward.

(a) Test Mesh

(b) Exponential

(c) Quadratic Inverse Distance

Figure 4.5: Mesh displacement diffusivity study using the mesh setup in (a), with the magnitude of grid point displacement based on diffusivity of (b) $\gamma = e^{-l}$, and (c) $\gamma = 1/l^2$.

(a) Exponential

(b) Quadratic Inverse Distance

Figure 4.6: Mesh quality at the tip of the flap with diffusivity (a) $\gamma = e^{-l}$, and (b) $\gamma = 1/l^2$.

As seen in Figure 4.5 and Figure 4.6, the quadratic inverse-distance mesh displacement diffusion method results in better mesh quality around the flap boundary as well as a smoother transition between the moving mesh points and the stationary ones. As such, the inverse-distance mesh motion solver was chosen to be used with the transient FIM solver.

4.3 Development of Solvers

4.3.1 The Steady-State Solver

The steady-state solver for the passive flap device was built around the automatic hex-mesh generating utility `snappyHexMesh` and consists of 5 different parts: the steady-state flow solver `simpleFoam`; the `Calculate_balance` subroutine; the `Angle_optimization` subroutine ; the automatic mesh generation utility `snappyHexMesh`; and the main computation loop. The OpenFOAM built-in solver and utility were written in C++ and utilized in the simulation without modification; the rest of the customized subroutines were written in C language. Old mesh files are removed at the beginning of every simulation loop, so that new mesh can be generated based on the updated airfoil and flap model that was defined in the STL model file.

Following the strategy outlined in Section 4.1, each iteration of the equilibrium angle seeking process began with mesh generation; as the flow domain is rectangular, an STL model file for the airfoil were generated for each of the angle of attack. The steady-state solver used to obtain the flow solution for each flap angle solves the incompressible Navier-Stokes equations

using the SIMPLE algorithm; it also produces output solution files and force reporting file at fixed intervals.

The `Calculate_balance` subroutine was developed to extract forces acting across the flap from the force reporting files, calculate the moment acting across the flap surfaces, and append the flap angle β and moment value into an output file that could be used by the optimization subroutine. The forces extraction and creation of output file were performed using standard file I/O functions in C, `scanf` and `fprintf`; the moment acting on the flap about the hinge is obtained by combining the components of forces as reported by the solver and gravitational force normal to the direction of the hinge axis.

The output file that contains the list of β and the corresponding moment about flap hinge is read in by `Angle_optimization` subroutine to find the equilibrium flap angle. The seeking of $\beta_{equilibrium}$ uses Newton's method for optimization to iteratively seek the flap angle β with zero moment acting on it; the subsequent β value produced by the iterative process is used by the subroutine to generate a new STL model file for the flap. The two β values used to obtain the initial solution for Newton's method are set such that the first value places the flap parallel to the flow direction, and the second value at 10° on top of the first β value.

The main computation loop assesses the difference between the current *beta* value and the one from the previous step, and determines if the termination condition is met; if not, the mesh files are deleted and replaced with new mesh generated using the airfoil and the new flap model files. Figure 4.7 shows an example of the different β value tested during the process of seeking $\beta_{equilibrium}$.

Figure 4.7: The series of flap angles tested, and the corresponding moments acting on the flap, throughout the equilibrium angle seeking process for a $c_f = 0.2c$ flap at $x_f = 0.8c$ on an SD8020 airfoil at $\alpha = 16^\circ$

4.3.2 Transient Fluid-Induced-Motion Solver

The transient fluid induced motion solver was built on top of the preexisting PIMPLE algorithm based dynamic mesh solver for user prescribed boundary motions. The motion of boundary patches in the stock solver is handled by special boundary condition, which as of OpenFOAM version 1.7.x is only capable of handling uniformly oscillatory and uniform directional motions; in either cases, the motion of the boundary patch is independent from the flow solution for the given time step. Although direct application of the existing boundary motion code is not possible, we were able to pin point the location to insert the flap motion code. The overall setup of the new transient FIM solver takes the following pseudo-code form, with the additional file for flap motion inserted into the solver via `#include` command:

```

#include 'readInput.H'
while (runTime.run())
{
    runTime++;
    #include 'rotateFlap.H'
    /* PIMPLE Loop */
    /* PISO Loop */
    #include 'calculateMoment.H'
}

```

The new FIM solver consists of two additional modules, `rotateFlap.H` and `calculateMoment.H`, inserted into `pimpleDymFoam`, the vanilla incompressible PIMPLE algorithm based transient flow solver with dynamics mesh support. As the OpenFOAM solvers are constructed based on a predefined C++ library tool set, the two aforementioned modules must be written in a fashion that the core solver understands. Although originally written for OpenFOAM version 1.5.x, and cannot be compiled for usage under version 1.6.x and 1.7.x due to difference in treatment of forces, the six degree of freedom volume of fluid solver developed by Erik Ekedahl[40] provided the necessary reference material to began the development of the two new modules. The governing equation for the flap motion and the code development for the two modules are explained in the following sections.

Governing Equation

Before divulging further into the code development process, we should first understand the equation that governs the motion of the flap. The motion of the flap itself is defined by Equation 4.35, which was also employed by the CFD code developed by T.U. Berlin researchers to solve for the passive flap problem [41]:

$$\theta_s \cdot \frac{d^2\beta}{dt^2} = -M_F - M_G, \quad (4.35)$$

where θ_s is the moment of inertia of the flap, and M_F & M_G are moment on the flap due to fluid and gravity, respectively. Equation 4.35 can be further rewritten into an explicit form solving for β as

$$\beta_{new} = -\frac{1}{2} \frac{(M_F + M_G)}{\theta_s} dt^2 + \dot{\beta}_0 dt + \beta_0, \quad (4.36)$$

which can take the discretized form

$$\Delta\beta = -\frac{1}{2} \frac{(M_F + M_G)}{\theta_s} \Delta t^2 + \dot{\beta}_0 \Delta t. \quad (4.37)$$

For simulation starting with the flap in a stationary position, the value for initial $\dot{\beta}$ would be zero, and M_F , M_G , θ_s , and Δt are all known values either predefined by the user or is calculated by the flow solver; for simplification, and to be consistent with the experimental data from water tunnel lab, the gravitational vector was set parallel to the axis of flap hinge.

Code for Flap Rotation

As mentioned in Section 4.2.4, changes in mesh and boundary patch position is handled by the vector field `pointDisplacement` in OpenFOAM, and must be updated to reflect the change in flap position as calculated by Equation 4.37. The translation of $\Delta\beta$ into change in coordinates of boundary points is accomplished by the construction of rotational matrix based on axis of rotation and change in flap angle from the initial ($t=0$) position. The changes

of boundary patch coordinated from the initial position is then updated into the `pointDisplacement` field using the following code:

```
forAllConstIter(labelHashSet, patchList_, iter)
{
    label patchi = iter.key();
    vectorField oldCell =
        mesh.C().boundaryField()[patchi];
    pointField oldPoint =
        mesh.boundaryMesh()[patchi].localPoints();
    // Rotate the patch by RotTen
    vectorField newCell =
        ((oldCell-CoR) & RotTen) + CoR;
    pointField newPoint =
        ((oldPoint-CoR) & RotTen) + CoR;
    // Cumulative point displacement
    pointDisplacement.boundaryField()[patchi] +=
        (newPoint - oldPoint);
    cellDisplacement.boundaryField()[patchi] +=
        (newCell - oldCell);
}
```

where `&` is the overloaded operator that performs dot product between different variables. The piece of code shown above also gave us an example of how object oriented programming in C++ helps simplified the coding process: instead of storing the boundary mesh position and corresponding variable values in separate arrays, OpenFOAM simply stores these values as separated member of the C++ object that defines `pointDisplacement` and `cellDisplacement`.

Code for Force Calculation

The development of the force extraction code was based on the OpenFOAM `forces` function object, which combines the pressure and viscous forces act-

ing on the boundary patch. The resulting code used for forces and moment calculation is presented below:

```
forAllConstIter(labelHashSet, patchList_, iter)
{
    label patchi = iter.key();
    vectorField Md =
        mesh.C().boundaryField()[patchi] - CoR;
    vectorField pf =
        mesh.Sf().boundaryField()[patchi]*
        p.boundaryField()[patchi];
    sumMoment += gSum(Md pf);
    vectorField vf =
        mesh.Sf().boundaryField()[patchi] &
        devRhoReffb[patchi];
    sumMoment += gSum(Md vf);
}
```

where `devReff()` is based on effective viscosity as calculated by the turbulent equations; `mesh.C()` field contains the center point position and `mesh.Sf()` contains the surface area of the user indicated patch. The resulting sum of moment about hinge axis is pass on to Equation 4.37 for the next time step.

To ensure that the motion solver is behaving correctly, and due to the lack of experimental data to validate against, simple test case was successfully conducted using the square cylinder used in the dynamics mesh solver test; the result showed that the flap would move, and that the movement of the flap is in the correct direction based on pressure difference across the flap surface.

Chapter 5

Simulation Setup

The purpose of the simulation study was to provide flow field information within the recirculation bubble and around the passive flap that is otherwise unavailable through experimental studies. By developing a solver that is capable of capturing the fluid-induced motion that is central to the lift enhancement ability of passive flaps, we could also use the solver as a development tool for the optimization of passive flap design. Simulations were conducted at $Re = 40,000$ using fluid properties presented in Table 5.1.

Table 5.1: Fluid Property

Fluid Properties	
Fluid Density	1.225 kg/m^3
Fluid Viscosity	$1.55 \times 10^{-5} \text{ m}^2/\text{s}$
Inlet Velocity	0.62 m/s

The chord length of the airfoil model in the simulations were set to $1m$ to simplify the post-processing process. Due to the difference between the capability and mesh generation process needed for the steady-state solver

(described in section 4.3.1) and transient FIM solver (described in section 4.3.2), the setup required to run the simulation is drastically different. The following sections documented the different mesh and solver setup needed for the 2D steady-state, 2D transient, and 3D transient simulations.

5.1 2D Simulation Setup

5.1.1 Steady-State Simulation Setup

Two dimensional steady-state simulation were conducted with the SIMPLE algorithm based solver `simpleFoam`, with all numerical schemes set to the default options: `steadyState` for time difference scheme; Linear Gauss integration scheme for the computation of gradients and divergence; the Laplacian is computed using Gauss discretization scheme with linear interpolation on the diffusion coefficient and unbounded-second order `corrected` scheme on surface-normal gradient. The flap is modeled with an internal plane with zero thickness and both side of the plane set to no slip condition; the gravitational vector is set in the direction normal to the simulation plane (y-direction in this case) and does not effect the simulation result. The automatic mesh generator was set to generate surface mesh size for $y^+ = 30$ with a growth rate of 1.2; as the mesh generator can only produce three-dimensional mesh, the side walls of the computational domain were set to `slip` boundary condition¹ so that it resembles a wing of infinite span, which produces identical results as 2D simulations. The dimensionless wall distance y^+ for the mesh

¹Boundary condition that enforces a fixed zero value for the wall normal vector, and zero-gradient to internal values for tangential vectors to the wall boundary.

was selected such that it falls within the y^+ range that the wall function in $k-\omega$ SST is capable of accurately model². The size of the fluid domain is setup to match the chord-to-test-section ratio from the water tunnel experiments by Schüter [15]. The simulation domain and the boundary patches are presented in Figure 5.1.

Figure 5.1: The model used in the steady-state 2D simulation with all boundary patches (except for the front and back faces) labeled.

The simulation models and mesh files for the steady-state simulations were generated by automatic mesh generator **SnappyHexMesh** from the OpenFOAM source code, and the airfoil model was generated using a 3D shell with the SD8020 airfoil profile in the x-z plane. The resulting mesh for SD8020 airfoil at $\alpha = 16$ using the automatic mesh generator is shown in Figure 5.2.

²The $k-\omega$ SST model uses standard wall function for $y^+ \geq 30$, while it can fully resolve the boundary layer flow if $y^+ \leq 1$. The turbulence model is unable to accurately simulate the boundary layer flow for $1 < y^+ < 30$.

(a) Overall Mesh

(b) Boundary Layer

Figure 5.2: Overall and detailed view of the mesh for SD8020 at $\alpha = 16$.

The lack of boundary layer mesh was due to the limitation of the automatic mesh generator, which is unable to generate boundary layer mesh on patches that intersect with more than one boundary patch; instead, the airfoil boundary mesh was generated using a growth function with starting size that corresponds to $y^+ \sim 30$ and a growth factor of 1.2. The lack of boundary layer cluster, however, lead to varying cell sizes along the surfaces that formed the airfoil model. In order to better understand the effect of

surface mesh size and the resulting flow solution, simulations were ran with clean airfoil configuration with minimum/maximum refinement level of 4/5, 5/6, and 6/7; this corresponds to an average y^+ value of 35, 26, and 10, respectively. The resulting lift curve is shown in Figure 5.3.

Figure 5.3: Mesh convergence study using clean airfoil configuration with different refinement setting in mesh generator **snappyHexMesh** at $Re = 40,000$.

The boundary conditions for each of the patches and the initial flow conditions used in the simulation are listed in Table 5.2 and 5.3.

Table 5.2: Boundary Conditions

Patch	Pressure	Velocity	k	ω
Inlet	zeroGradient	fixedValue	fixedValue	fixedValue
Outlet	atmospheric	no backflow	zeroGradient	zeroGradient
Airfoil	zeroGradient	no-slip	wall function	wall function
Flap	zeroGradient	no-slip	wall function	wall function
Side Walls	zeroGradient	slip	slip	slip

Table 5.3: Initial Conditions

Variable	Values
Pressure	0 <i>Pa</i> Gauge
Velocity	(0.62, 0, 0)
k	0.01215
ω	0.20125

Simulations were ran for 1000 time steps to ensure flow convergence for each of the configurations as the main subroutine cycles through multiple β values in order to find the equilibrium, and optimal, flap angle that the pressure difference due to recirculation bubble can sustain.

5.1.2 Transient Simulation Setup

The transient simulation solver uses similar numerical scheme to the ones used in the steady-state solver, with the exception of additional Laplacian schemes settings required by the mesh motion solver and the Euler scheme for time difference. The mesh used in the simulation was generated using Ansys GAMBIT mesh generator, and the flow domain was modeled with a C-grid with the center point of the airfoil at the center of a $6m$ radius semi circle; the chord length of the airfoil is set to $1m$. The coordinate of the airfoil was imported into GAMBIT as vertices, which were connected via non-uniform rational B-spline to form the 2D airfoil model. The flap was modeled as a zero-thickness internal surface, and the direction of gravitational force was set in the parallel direction to the axis of rotation for the passive flap (the z-direction); the flap density was set to $2000kg/m^3$ with a flap “thickness” of $5 \times 10^{-3}m$, a value that was used in conjunction with the flap length to

calculate the sectional moment of inertia of the flap; both of the values are adjustable in the configuration file for the solver. The moment of inertia was in turn used in the FIM solver to calculate the flap displacement via Equation 4.37. The two dimensional simulation domain and the airfoil model is shown in Figure 5.4.

Figure 5.4: The model used in the transient 2D simulation with all boundary patches labeled.

Unlike the semi-two dimensional flow domain used in the steady-state simulation that could not create boundary layer mesh due to the limitation of the automatic mesh generator, the GAMBIT pre-processor allows the transient simulations to be conducted on a two-dimensional grid with user defined boundary layer mesh around the flap and the airfoil. As OpenFOAM

was developed as a continuum mechanics solver, all 2D simulation models has to be extruded in the normal direction to the simulation plane for one cell width; the side-walls are set to **empty** boundary condition so the solver recognized that the simulation is for 2D flow field, and calculation would not be carried out for the third direction (z-direction in this case). The boundary patches used in the 2D transient simulation have identical name and boundary type as the ones used in the steady-state simulation in order to simplify the setup process for the OpenFOAM cases, which requires modification of the boundary condition files through command line text editor such as VIM for each of the variables.

Transient simulations were conducted using both a $y^+ = 30$ and a $y^+ = 1$ boundary layer mesh setup, with the former modeling the boundary layer flow using the logarithmic wall function while the later was able to fully resolve the boundary layer flow without the need for modeling with wall functions. The time required in order to obtain a single usable simulation data point is typically 24 hours for the coarse grid, and up to a four days for the dense grid, using 16 CPUs on two computation nodes to the simulation. The overall 2D mesh setup as well as the boundary layer mesh around the airfoil and flap patches were shown in Figure 5.5 and Figure 5.6; the cell counts for the two mesh setups are presented in Table 5.4.

Table 5.4: Cell Counts and Dimensionless Wall Distance for Fine and Coarse Meshes

Mesh	Quad/Hex Cells	Tri/Prism Cells	Average y^+
Fine 2D Mesh	14800	194	0.1765
Coarse 2D Mesh	2075	53	28.5879

(a) Overall Mesh

(b) Boundary Layer

Figure 5.5: Overall and detailed view of the mesh for SD8020 at $\alpha = 16$ with $y^+ = 30$ boundary layer mesh.

(a) Overall Mesh

(b) Boundary Layer

Figure 5.6: Overall and detailed view of the mesh for SD8020 at $\alpha = 16$ with $y^+ = 1$ boundary layer mesh.

The boundary layer mesh around the airfoil and flap surfaces were constructed using boundary layer tools available in GAMBIT: the starting grid size was calculated based on the the corresponding y^+ value; a mesh growth rate of 1.2 was chosen based on past experience; the number of enforced mesh layers were set to 4 and 20 for the coarse and fine mesh, respectively, to ensure continuity between the wall boundary mesh and the free stream mesh. The flap was modeled with an initial $\beta = 0$ with respect to the free stream flow to ensure there is enough space for mesh generation between the flap and the upper surface of the airfoil. The triangular region between the flap and the airfoil was meshed using unstructured tri-mesh to ensure the mesh quality, and is the only portion of the simulation regime consists of unstructured mesh; the rest of the simulation regime were discretized with quad-mesh using "Map" meshing scheme and a maximum growth rate of 1.2. Due to the initial orientation of the flap surface, and to simplify the process of mesh generation, the airfoil models were rotated to the desired angle attack before mesh generation for each simulations instead of modifying the inlet flow angle³.

Each simulations were initialized using potential flow solver followed by 10 seconds⁴ of flow simulation with the flap held stationary; this was done to ensure that the separation bubble is fully developed so that the flap is not forced down by the initial attached flow field predicted by the potential flow solver. The flow solution is then read into the FIM transient solver and simulated for an additional 40 seconds of flow; the displacement of the mesh

³Due to the setup of the OpenFOAM file structure

⁴in simulation.

points within the simulation domain are calculated at the beginning of each time step using the dynamic mesh solver described in Section 4.2.4, and all internal mesh points displaced accordingly instead of re-meshing the entire domain. The size of the simulation time-step was chosen automatically by the OpenFOAM source code based on prescribed maximum Courant number, in this case 0.01s. However, smaller time-step size can be enforced if instability occurs.

Additionally, in order to prevent overly skewed meshes between the flap and the airfoil that would result in poorly resolved flow solution and introduce instability into the simulation process, a β limit was introduced such that there is at least 5° separation between the flap and the airfoil; if the flap is lowered to that β with a sustained moment pushing it further downward, the flap configuration is assumed to have a fully collapsed flap that produce no additional contribution to lift. For the cases where the flap is assumed to have collapsed, producing no additional lift, the lift data of the same airfoil with the same mesh, but without the flap patch declared as a `wall` boundary patch, is used.

5.2 3D Simulation Setup

The 3D finite wing simulations were conducted to provide visualization and flow analysis capability needed for better understanding of the observations made in the finite wing wind tunnel study. As the instruments in the wind tunnel is only capable of measuring forces acting on the models, it was hoped that the more detailed flow information yielded from the numerical simula-

tion could help us better understand the flow field surrounding the passive flap and how it influences the design parameters that resulted in the optimal passive flap configuration. Following the 2D transient simulations, the setup of which were described in Section 5.1.2, we were confident that the choice of turbulence model and the transient FIM code could accurately simulate the flow field surrounding the airfoil and the flap, though further testing is required to ensure the compatibility of the solver with the additional boundary condition types needed for a 3D simulation.

The three-dimensional model employed the symmetry plane boundary condition in order to simulate the whole finite-span wing with half of the required mesh points. The cross sectional profile of the simulation domain and model setup for the three-dimensional simulation is identical to the one employed in the 2D transient simulation: the portion of the domain where the wing resides was extruded over its half-span directly from the aforementioned 2D mesh, with the end of the wing capped with a single perpendicular surface; the end-surface is subsequently meshed with triangular unstructured mesh, and the mesh was further extruded for another $b/2$ to complete the construction of the simulation domain. The hex-mesh portion of the fluid domain were extruded via “Map” scheme, while the tri-mesh portion of the domain was extruded via “Cooper” scheme as prisms. The geometry of the simulation model with the individual patches, with the exception of the **left** patch which is transparent, distinctively colored is presented in Figure 5.7.

Figure 5.7: The model used in the 3D transient simulations. All patch names remains the same as the 2D transient case with the exception of the front (left) and back (right) surface of the domain.

The span of the flap can be re-sized by extruding the mesh in segments, which introduces internal lines and surface that divide the flap surface into sections; the flap span is determined by choosing the desired internal surfaces in boundary condition selection, while the rest of the surface sections are assimilated into the internal mesh during the mesh export process.

Boundary layer mesh of $y^+ = 1$ was chosen based on the results from 2D transient simulations, though a single coarse ($y^+ = 30$) mesh was constructed for $\alpha = 16$ in order to ensure that the lateral (front and back) boundary patches are behaving as intended for the FIM solver. The boundary mesh over the wing section is identical to the one presented in Figure 5.6, while the x-y cross-section for the rest of the mesh is shown in Figure 5.8.

Figure 5.8: The close-up view of x-y cross-section mesh for the 3D mesh portion that does not include the wing model. Prism type mesh were used to discretize the space that formerly defined the wing model.

Mesh discretization in z-direction are controlled by two separated size-functions, both defined by the first length scheme $\sum_{i=1}^n R^{i-1} = L/l_1$, where R is successive ratio, n is the number of mesh intervals, L is the length of the edge to be discretized, and l_1 is the size of the first mesh point. The l_1 value for meshed edges starting from the wingtip towards the root (negative z direction) is $0.01m$, with $n = 20$; for mesh points starting from the wingtip towards the edge of the fluid domain in the positive direction, the l_1 is set to $0.001m$ (approximately equal to $y^+ = 3$ from the end-surface of the wing), with $n = 30$. The distribution of mesh points in the z-direction is shown in Figure 5.9.

Figure 5.9: Distribution of mesh points in the z-direction.

Chapter 6

Airfoil Simulation Studies

The primary objective of the two dimensional study of passive flap application on an airfoil is to examine the ability of the turbulence model to replicate the recirculation flow: the successful application of computation method for the low Reynolds number passive flap opens the possibility of faster optimization process for the configuration of passive flap for the particular wing or airfoil design. The accuracy of the turbulence model in simulating flow separation and reattachment is assessed via comparison of the predicted lift coefficient with the experimental results from the NTU water tunnel [15]; as the water tunnel data was collected with the span of the model in vertical direction, the direction of gravitational force is aligned with the axis of rotation of the flap in the 2D simulations. The successful application of the CFD simulation should allow us to examine the recirculation and turbulent separation flow field around the passive flap in finer detail comparing to what was possible in an experimental setting. The converged velocity contour plots will be provided for converged steady-state cases, while the same

plots for the transient case will be displayed with time-stamp.

6.1 Steady-State Simulation

In order to ensure the accuracy of stall prediction from the turbulence modeling, simulations were carried out to obtain the simulated lift curve for airfoil without the passive flap attached¹. The resulting lift data is plotted against experimental data from Selig et al. [17] in Figure 6.1.

Figure 6.1: Lift Curve for clean SD8020 airfoil from the simulation compared to experimental results.

In addition to the lift curves for each of the configurations, velocity contours of the clean airfoil case will be presented side by side with the ones equipped with passive flap device in order to illustrate the effect of the passive flap on the separation flow field.

¹This baseline configuration will be referred to as “Clean Airfoil” setup in the following sections.

6.1.1 $x_f/c = 0.8$, $c_f/c = 0.2$ Flap Configuration

The lift curve of the $x_f/c = 0.8$, $c_f/c = 0.2$ passive flap configuration comparing to the experimental data at $Re = 40,000$ is presented in Figure 6.2.

Figure 6.2: Lift Curve for $x_f/c = 0.8$, $c_f/c = 0.2$ from the simulation compared to experimental results.

The lift curve comparison showed that the CFD simulation predicted lift generation of SD8020 airfoil with $x_f/c = 0.8$, $c_f/c = 0.2$ passive flap attached matches closely with the water tunnel experimental data [15] for majority of the data points, which was subjected to $\sim 10\%$ error in measured values. Comparing to the experimental data, the simulation result appeared to have over-predicted the boundary layer retention until $\alpha = 14^\circ$, at which point the lift curve fell back inline with the clean airfoil data, indicating complete flow separation over the airfoil.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.2; x_f/c = 0.8$, Velocity

Figure 6.3: Velocity contour of clean airfoil and $c_f/c = 0.2; x_f/c = 0.8$ flap at $\alpha = 10^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.2; x_f/c = 0.8$, Velocity

Figure 6.4: Velocity contour of clean airfoil and $c_f/c = 0.2; x_f/c = 0.8$ flap at $\alpha = 12^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.2; x_f/c = 0.8$, Velocity

Figure 6.5: Velocity contour of clean airfoil and $c_f/c = 0.2; x_f/c = 0.8$ flap at $\alpha = 14^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.2; x_f/c = 0.8$, Velocity

Figure 6.6: Velocity contour of clean airfoil and $c_f/c = 0.2; x_f/c = 0.8$ flap at $\alpha = 16^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.2; x_f/c = 0.8$, Velocity

Figure 6.7: Velocity contour of clean airfoil and $c_f/c = 0.2; x_f/c = 0.8$ flap at $\alpha = 18^\circ$.

Influences of the passive flap on the recirculation flow field is especially noticeable in Figure 6.4 and 6.5, where the turbulent separated flow behind the airfoil was significantly reduced as recirculation zone was formed between the flow detachment point and the passive flap. On the other hand, Figure 6.6 and 6.7 showed that the passive flap became ineffective as the separation flow regime overtook the flap's range of motion, and hence influence.

6.1.2 $x_f/c = 0.6$, $c_f/c = 0.4$ Flap Configuration

Further simulations were conducted using the $x_f/c = 0.6$, $c_f/c = 0.4$ flap configuration with the expectation that the larger flap size would produce a more distinctive visual difference in the flow field; we also hoped to find out why the larger flap size resulted in a similar lift enhancement as the $x_f/c = 0.8$, $c_f/c = 0.2$ configuration [15].

Figure 6.8: Lift Curve for $x_f/c = 0.6$, $c_f/c = 0.4$ from the simulation compared to experimental results.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.4; x_f/c = 0.6$, Velocity

Figure 6.9: Velocity contour of clean airfoil and $c_f/c = 0.4; x_f/c = 0.6$ flap at $\alpha = 10^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.4; x_f/c = 0.6$, Velocity

Figure 6.10: Velocity contour of clean airfoil and $c_f/c = 0.4; x_f/c = 0.6$ flap at $\alpha = 12^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.4; x_f/c = 0.6$, Velocity

Figure 6.11: Velocity contour of clean airfoil and $c_f/c = 0.4; x_f/c = 0.6$ flap at $\alpha = 14^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.4; x_f/c = 0.6$, Velocity

Figure 6.12: Velocity contour of clean airfoil and $c_f/c = 0.4; x_f/c = 0.6$ flap at $\alpha = 16^\circ$.

(a) Clean Airfoil, Velocity

(b) $c_f/c = 0.4; x_f/c = 0.6$, Velocity

Figure 6.13: Velocity contour of clean airfoil and $c_f/c = 0.4; x_f/c = 0.6$ flap at $\alpha = 18^\circ$.

6.1.3 Discussion

The lift curve for $x_f/c = 0.6$, $c_f/c = 0.4$ simulation showed better retention of attached flow comparing to the clean airfoil configuration as well as the $x_f/c = 0.8$, $c_f/c = 0.2$ past stall angle, an observation that can be explained by observing the difference in separation flow field between Figure 6.6 and 6.12: the former showed a completely separated flow field over the upper airfoil, with the flap lying within the turbulence wake flow and have minimal impact on the size of flow separation; the later showed an intact recirculation flow field between the initial flow separation point and the flap, with the turbulence flow separation limited to the area behind the flap. The contour plots confirmed that the formation of recirculation bubble between the leading edge and the passive flap via flow reattachment contributed towards the generation of lift, albeit at a lower efficiency; it also indicated that the lift curve should converge with the clean airfoil lift curve after the recirculation bubble bursts.

The stream-trace analysis of the separation flow field with and without the passive flap installed is presented in Figure 6.14 and 6.15, showing the difference between the recirculation flow field: comparing to the clean airfoil, the passive flap created a smaller but stronger recirculation flow between the separation point and the tip of the flap, which redirected the free stream flow so that the flow field is reattached onto the flap; the redirection of flow also resulted in a smaller turbulence wake profile. The stream-traces were integrated using Runge-Kutta 4-5 scheme.

(a) Clean Airfoil

(b) $c_f/c = 0.2; x_f/c = 0.8$

Figure 6.14: Comparison of stream-trace ribbon plot, from the 2D steady-state case, of the recirculation zone at $\alpha = 14^\circ$ with and without the flap.

(a) Clean Airfoil

(b) $c_f/c = 0.2; x_f/c = 0.8$

Figure 6.15: Comparison of stream-trace ribbon plot, from the 2D steady-state case, of the free stream flow at $\alpha = 14^\circ$ with and without the flap.

The simulation results showed good accuracy with the pre-stall lift prediction, but tend to over-predict the lift enhancement ability of the passive flap post-stall for both of the flap configurations even if we take into account of the reported 10% uncertainty in the experimental results. The small discrepancy between the experimental data and the simulation results could be due to the inability of the automatic mesh generator to properly “snap” boundary points onto the airfoil geometry defined by the STL model, creating bumps shown in Figure 6.16 that could form vortices and turbulence that would not exist on an accurately generated airfoil model; it could also be due to the lack of boundary layer mesh, which led to less than perfect boundary flow prediction due to the irregular sizing of cells closest to the airfoil surface (as shown in Figure 6.17.)

Figure 6.16: Model of SD8020 airfoil showing the bumps (circled in red) where the mesh generator failed to properly snap the mesh points onto the airfoil model.

(a) Without Grid Lines

(b) With Grid Lines

Figure 6.17: SD8020 airfoil simulation at $\alpha = 16$ showing the boundary layer flow prediction with and without the grid line. The lack of uniform boundary layer cluster caused the irregularity in boundary flow.

Although not without flaws, the steady-state simulation gave us an idea of how the design variables c_f and x_f would affect the lift enhancement property of the passive flap device. The problems with mesh generation as well as the inability of the solver to be adapted for URANS/LES simulation, though, led to the development of the transient solver.

6.2 Transient Simulation

Various modeling and simulation problems encountered while conducting steady-state simulations, as discussed in Section 6.1.3, prompted the development of the transient FIM solver. Like the steady-state solver, the results from transient simulation were compared to the experimental data from the water tunnel study [15] in order to ensure that the turbulence models and the FIM codes are behaving correctly.

6.2.1 Simulation Results

The transient simulations were conducted with the baseline setup of $x_f/c = 0.8$ and $c_f/c = 0.2$, with the aim of amassing enough data for comparison with the steady-state simulation and water tunnel experimental results; the configurations were chosen for the availability of both 2D and 3D experimental data, as the 3D meshes were directly extruded from the 2D ones in order to expedite the pre-processing procedure. The lift curves from all three 2D transient simulations, conducted with different mesh setup and turbulence models, are presented along with the corresponding experimental data in Figure 6.18.

Figure 6.18: Lift curves from the 2D transient simulations using different mesh density and turbulence models.

Initial simulations were conducted using $k-\omega$ SST turbulence model for both coarse and dense mesh setup, with additional simulation conducted with Spalart-Allmaras turbulence model using the coarse mesh setup later for trouble shooting when the previous simulations does not return accurate flap movements. The coarse mesh allows the trouble shooting simulations to be completed in two days instead of a week. The Spalart-Allmaras simulation results falls along a gentle curve extending from the clean airfoil data line; the results from the $k-\omega$ SST simulation with the coarse mesh follows closely to the lift curve from the Spalart-Allmaras simulation; the $k-\omega$ SST simulation with the fine mesh setup provided results the most closely matched to the experimental results. The simulations were limited to $12^\circ < \alpha < 20^\circ$ as we are mainly interested in the performance of the flap between $\alpha = 12^\circ$, the original stall angle, and $\alpha = 16^\circ$, the onset of turbulent wake that signify the

stalling of airfoil with passive flap.

The difference between the results from the coarse mesh and the fine mesh setup could be caused by the difference in mesh resolution within the recirculation zone, as shown in Figure 6.19 and 6.20 above, as only the first three layers of the boundary mesh lies within the recirculation flow field in the simulations conducted using the coarse mesh setup. The lack of mesh density within the recirculation zone led to inaccuracy in the prediction of boundary layer flow, even though wall function in $k-\omega$ SST should provide an accurate boundary layer profile with the dimensionless wall distance set to $y^+ = 30$; the lack of mesh density could also affects the prediction and resolution of turbulence eddies caused by flow separation, which can be seen in Figure 6.19 where the velocity profile from the fine mesh setup shows turbulence eddies and flow features within the recirculation bubble that did not present in the coarse mesh simulations. Another possible explanation for the over-prediction of lift by the coarse mesh setup could be due to the nature of the wall function, which models the average boundary flow velocity based on the distance from the wall, and lead to inaccurate prediction of flow velocity within the recirculation region. On the other hand, simulation results from the fine-mesh setup, though producing one less data point at $\alpha = 20$ comparing to the coarse mesh simulations, follows the lift curve from the experimental data pretty closely, and showed that the recirculation zone is not as stable and stationary as we first assumed. Instead of a single recirculation bubble within the space between the free stream flow, the airfoil, and the flap, the recirculation zone is formed by a string of recirculation bubbles initiated at the flow separation point (Figure 6.21 & 6.22).

(a) Fine Mesh, $k-\omega$ SST

(b) Coarse Mesh, $k-\omega$ SST

(c) Coarse Mesh, Spalart-Appmaras

Figure 6.19: Comparison of transient flow solutions at $t = 10.1s$, with the airfoil angled at $\alpha = 16^\circ$.

(a) Fine Mesh, k- ω SST

(b) Coarse Mesh, k- ω SST

(c) Coarse Mesh, Spalart-Appmaras

Figure 6.20: Comparison of pressure fields from the transient simulations at $t = 10.1s$, with the airfoil angled at $\alpha = 16^\circ$.

(a) $t=10$

(b) $t=10.1$

Figure 6.21: Transient flow solution of SD8020 at $\alpha = 16$ with flap configuration of $x_f/c = 0.8$ and $c_f/c = 0.2$.

(a) $t=10.2$

(b) $t=10.3$

Figure 6.22: Transient flow solution of SD8020 at $\alpha = 16$ with flap configuration of $x_f/c = 0.8$ and $c_f/c = 0.2$.

The unsteady formation of recirculation bubble could explain for the oscillatory behavior of the flap observed in the experimental studies, though a more detailed study of the oscillatory motion is currently not possible: although the current solver contains the β values in system memory and write it into the solver's log file; however, the size of the log file ($\sim 14GB$ for 10s in simulation time) and restriction in storage space on the super computer meant that the log file is constantly overwriting itself. Additionally, the wind tunnel facility was not equipped for the recording of β and flap oscillation frequency data for passive flap on rectangular wing with SD8020 profile, preventing the verification of simulation data.

6.2.2 Discussion

The coarse grid simulations from both $k-\omega$ SST and Spalart-Allmaras studies showed that the $y^+ = 30$ boundary layer resolution is insufficient to accurately capturing the flow separation, and could inevitably cause the turbulence model to switch into high Reynolds number wall treatment mode; the mesh also does not provide enough cell within the recirculation region in order to capture the recirculation flow. The two different turbulence model used in the coarse mesh simulation also allowed us to see the difference between flow separation point as a result of the different boundary-layer treatments between the models (Figure 6.19(b) & 6.19(c).) It should be noted that the simulation studies were limited to $12 < \alpha < 20$ as we are primarily concerned with the separation of flow from the upper surface of the airfoil which leads to the deployment of the flap; as angle of attack increases

past the flap deployment angle, the lift would increase until the recirculation bubble between the free stream flow and the airfoil/flap is no longer sustainable and resulted in secondary stall as observed in the experimental studies.

On the other hand, the numerical difficulties encountered by the 2D SST simulations were mainly due to the excessive build up of k and ω in the corner between the lower surface of the flap and the upper surface of the airfoil; the same turbulence variables appears to diffuse out in the 3D simulations and does not pose as much trouble to the simulations. The build-up of turbulence kinetic energy and specific dissipation could be due to the sudden change in boundary layer density at the end of the flap, where the $y^+ = 1$ boundary layer mesh at the upper surface of the flap met with the coarse triangular mesh at the corner between the flap and the airfoil. Since the flow velocity and pressure difference within the triangular space between the flap and the airfoil is small, the build-up of the turbulence variables does not have a discernible impact on the overall lift data; it does, however, severely affect the stability of the simulation if the value of k and ω exploded locally (as shown in Figure 6.23). The aforementioned problems regarding the simulation stability could be correctable by modifying the mesh composition to form a smoother transition in mesh density across the flap or by employing the Spalart-Allmaras turbulence model with modified constants that could accurately capture the flow separation.

(a) Turbulence Kinetic Energy

(b) Specific Dissipation

Figure 6.23: Turbulence kinetic energy and specific dissipation fields at $t=1.2$ from unstable simulations with insufficiently small time step size.

While the collection of experimental drag data with the instruments available is problematic, we were able to calculate the drag data of airfoil with passive flap with relative ease. It should be noted that the drag data collected were based on no-slip wall condition and zero flap thickness, and do not account for the effects of surface friction, surface imperfection, and thickness of the flap on drag; due to the lack of experimental drag data at this Reynolds number from the literature, and the lack of accurate drag data from our own experiments, the drag data from simulation was not validated. As it was established that the simulation results with the coarse mesh are unreliable, Figure 6.24 below included only the lift over drag data for the clean airfoil configuration and $c_f = 0.2$, $x_f = 0.8$ data from the fine mesh setup.

Figure 6.24: The lift over drag ratio versus angle of attack plot from the clean airfoil configuration and airfoil with flap of $c_f = 0.2$, $x_f = 0.8$ configuration.

The figure above showed that airfoil with flap has better L/D performance directly past the original stall angle of the airfoil, though the improvement rapidly diminishes as the airfoil-flap combo itself stalls. Another interesting data set to consider is the effect of flap on the pitching moment of the airfoil; the coefficient of moment about mid-chord is presented in Figure 6.25.

Figure 6.25: The moment coefficient versus angle of attack plot from the clean airfoil configuration and airfoil with flap of $c_f = 0.2$, $x_f = 0.8$ configuration.

Judging from the figure above, the addition of passive flap appears to reduce the overall pitching moment; the existence of recirculation bubble might have caused enough alteration to the chord-wise force distribution along the upper surface of the airfoil to change the pitching moment. Like the lift over drag ratio, however, additional data is required to analyze the full effect of the flap on pitching moment.

Chapter 7

Finite Wing Studies

The estimation of lift contribution by passive flaps in real life applications are complicated by the inclusion of forces from the third dimension, namely the contribution from the wing-tip vortex. This chapter aims to discuss the effect of non-uniform lift distribution on passive flap performance equipped with various flap positions and configurations. The following sections document the lift contribution from the passive flaps configured with different flap length (c_f), flap span (b_f), and flap position (x_f)¹.

7.1 Experimental Results

The following sections document the lift curves obtained via wind tunnel study, with the resulting data sorted to highlight the effect on the lift by each of the factors involved in flap design. The results were previously published in Comptes Rendus Mécanique by the author [42].

¹As LibreOffice is unable to produce plot legend containing fractions and super/subscript, c_f/c will be denoted as c , x_f/c denoted as C , and b_f/b as S .

7.1.1 The Effect of Flap Span b_f

The effective angle of attack increases from the wing-tip as the downwash propagates inward due to finite wing effect. The resulting non-uniform lift distribution leads to different level of flow separation over the span of the wing that requires different degree of passive flap deflection for optimal lift enhancement. Considering the deviation in sectional lift coefficient is relatively small close to the center of the wing, it could be reasoned that a shorter flap (with one edge flushed with the root of the wing) would result in less interference with the still attached flow closer to the tip of the wing. The purpose of the following experiments was to determine the right balance between reducing interference and obtaining sufficient lift enhancement. The lift curves from the experiments are presented in Figure 7.1, 7.2, and 7.3.

The experimental results from flaps with $b_f = 1$ and $b_f = 0.8$ indicated that the reduction in flap span indeed reduces the effect of downwash on the lift enhancement property of the passive flap device. The results from $b_f = 0.6$ and $b_f = 0.4$, on the other hand, demonstrated that excessive reduction in flap span would render the device ineffective. From the experimental results, we can conclude that the optimal flap span for a single passive flap is for it to span over the portion of the wing where deviation of sectional lift is small; this would suggest that flap design with multiple section in the span-wise direction should be able to better capture the localized flow separation. To test the effect of the passive flap device with more than one sections, an additional flap configuration were included in Figure 7.3 with an inner flap span of $60\%b$ and an outer flap span of $40\%b$.

(a) $0.3c/0.7C$

(b) $0.3c/0.6C$

Figure 7.1: Lift curves of passive flaps with different flap spans at $Re = 4 \times 10^4$.

(a) 0.3c/0.5C

(b) 0.1c/0.9C

Figure 7.2: Lift curves of passive flaps with different flap spans at $Re = 4 \times 10^4$.

(a) 0.1c/0.8C

(b) 0.1c/0.7C

Figure 7.3: Lift curves of 0.1c passive flaps at $Re = 4 \times 10^4$.

(a) 0.1c/0.6C

(b) 0.1c/0.5C

Figure 7.4: Lift curves of 0.1c passive flaps at $Re = 4 \times 10^4$.

(a) 0.2c/0.8C

(b) 0.2c/0.7C

Figure 7.5: Lift curves of 0.2c passive flaps at $Re = 4 \times 10^4$.

(a) 0.2c/0.6C

(b) 0.2c/0.5C

Figure 7.6: Lift curves of 0.2c passive flaps at $Re = 4 \times 10^4$.

The composite flap configuration yielded better post-stall recovery comparing to the single flap setup, as the additional lift contribution from the outer flap having a positive effect on the overall lift generation. On the other hand, previous studies by Patone & Müller [11] and Kernstine et al. [14] reported that multi-sectional passive flap configuration would adversely affect the overall lift generation. The conflicting results suggest that although dividing passive flap into sections is beneficial to the reattachment of local flow separation, the cross-flow between the different flap segments and the vortex created by the edges would disturb the flow field on the upper surface of the airfoil, resulting in the lost of lift.

7.1.2 The Effect of Flap Position x_f

As the activation of passive flap device is triggered by flow separation, it could be assumed that the optimal positioning of the flap's x_f is dependent on the position of flow separation point for the airfoil. Previous study by Bechert et al. [12] suggested that the passive flap should be positioned such that the trailing edge of the flap is slightly ($\sim 0.2\%c$) upstream from the trailing edge of the airfoil based on the assumption that flow separation travels upstream, i.e. trailing edge stall; on the other hand, experiments by Kernstine et al. [14] indicated that the flap should be positioned as close to the leading edge as possible without compromising the upper surface curvature of the wing profile. Experiments were carried out with flaps with chord size of $c_f/c = 0.3$ and span of $b_f/b = 1.0$ and $b_f/b = 0.8$; the recorded lift data are presented in Figure 7.7.

(a) $b_f/b=1.0$

(b) $b_f/b=0.8$

Figure 7.7: Lift curves of $c_f/c = 0.3$ passive flaps at different chord-wise position at $Re = 4 \times 10^4$.

Although the design of SD8020 airfoil indicated that a leading edge stall pattern should be observed, the experimental results showed very little difference between $x_f = 0.6$ and $x_f = 0.7$, while the $x_f = 0.5$ setup produced the least lift enhancement. The possibility of flow disruption due to the thickness of the flap had been ruled out, as experiments conducted with taped down flaps performed identically to the unmodified wing. The experimental data also show that positioning the flap too close to the point of flow separation might not be ideal, as the $x_f = 0.5$ flap failed to deploy even at very high angle of attack; this could be due to the smaller space available for the recirculation bubble to form once the flap is deployed, producing insufficient pressure difference across the flap to remain lifted. No experiments were conducted for $x_f < 0.5$ as the rigid flap was unable to conform to the curvature of the wing, and would alter the lift characteristic of the wing regardless of the flap being deployed or not.

7.1.3 The Effect of Flap Chord c_f

As the passive flap is activated by the recirculation flow, its movement is constrained to within the separation zone. The confinement to the separation zone means that larger flap size could completely cut the flow separation in two, forming two smaller but stronger recirculation flow fields. It would be interesting to compare the performance to that of a smaller flap with its partially separated flow field. It would be interesting to find out if the recirculation flow diversion from a smaller flap is capable of achieving the same lift increase of a larger flap. Figure 7.8 shows the results.

(a) $0.7x_f/c$

(b) $0.6x_f/c$

Figure 7.8: Lift curves of $0.8b_f/b$ passive flaps with various flap chord-length at $Re = 4 \times 10^4$ compared with data without the flap. The flaps are positioned at $0.7x_f/c$ for 7.8(a) and $0.6x_f/c$ for 7.8(b).

The experiments showed that the $c_f/c = 0.3$ flap exhibits significantly better lift enhancement comparing to the $x_f/c = 0.1$ and $x_f/c = 0.2$ flaps; the $x_f/c = 0.1$ flap very rarely respond to the recirculation flow, and when it does showed very little effect on lift generation. The difference in lift recovery might be due to the difference in the size of the resulting recirculation flow: larger flap diverts more airflow behind it, causing a stronger recirculation flow that is more capable of pulling the free stream flow down and create more lift. It should be noted that the complete reversal (tipping-over) of the flap was observed at very high angle of attack, and would result in hysteresis in lift curve as the flipped flap is unable to contribute to the lift generation; the reversal of the flaps cannot be self-corrected until flow reattachment occurred. In order to counter problems caused by the flap reversal, device that limits the movement range of the passive flap should be used in a fashion that does not interfere with the normal function of the flap.

7.1.4 The Effect of Reynolds Number

The actual operation of the MAV would not be confined to a single Reynolds number, so a good lift enhancement device should maintain its performance throughout the operation Reynolds number range. The following experiment was designed based on the assumption that takeoff and landing of the MAV occur at $Re = 4 \times 10^4$, which is at the minimal speed required to maintain lift. Kernstine et al. [14] reported a positive effect of Reynolds number on lift enhancement from their tests at $Re = 1.66 \times 10^5$ and $Re = 4.53 \times 10^5$, further testing on the subject was limited by their flap material. The current

experiments were conducted at $Re = 4 \times 10^4$, $Re = 5 \times 10^4$, and $Re = 6 \times 10^4$ using $c_f/c = 0.2$ and $c_f/c = 0.3$ flap with flap span $b_f/b = 0.8$; the Reynolds numbers were chosen based on the flight envelope of the MAV and operational limit of the wind tunnel² as well as the structural limit of the wing model. The results are reported in Figure 7.9 & 7.10 below.

(a) Reynolds number comparison

(b) $Re = 60000$

Figure 7.9: Lift curves of wings with passive flap at various Reynolds number compared to that of clean wings at the same Reynolds number.

²Due to structural concern with the retractable hanger door, the wind tunnel test section speed is limited to $12m/s$.

(a) $Re = 50000$

(b) $Re = 40000$

Figure 7.10: Lift curves of wings with passive flap at various Reynolds number compared to that of clean wings at the same Reynolds number.

The plot demonstrated the effect of Reynolds number on the maximum lift coefficient of the clean airfoil, which caused a slightly delayed post-stall recovery at higher Reynolds number for flaps configured with the same c_f and x_f values. The Reynolds number experiment showed mixed results, while higher Reynolds number generates a higher lift coefficient post-stall, the relative increase in lift is less. We associate the lower lift increase at $Re =$

6×10^4 to the delay in flow-separation, and hence the delay in passive flap deployment; the pressure distribution on the low Reynolds number airfoil could also have an effect on the passive flap performance at higher Reynolds number.

7.1.5 Drag Data

While drag data was collected along side with the lift data in wind tunnel experiments, the accuracy of the drag force measurement³ is limited by the resolution of the instrument⁴. The drag data of finite wing with SD8020 profile is compared to the same wing with passive flap of $c_f = 0.2$, $x_f = 0.8$, and $b_f = 1.0$ configuration attached is presented in Figure 7.11 below.

Figure 7.11: Wind tunnel experiment on finite wing with drag data collected from clean SD8020 profile and finite wing with passive flap of $c_f = 0.2$, $x_f = 0.8$, and $b_f = 1.0$ configuration.

As seen in the figure above, the drag force collected prior to $\alpha = 10$ are

³Does not exceed 0.5N in our experiments.

⁴Error margin of $\pm 0.05N$

mostly negative while in theory the drag force should always be positive; the drag data does return numbers very close to zero assume full error margin, though. The negative drag reading could be due to uncertainties and noises in force measurement, but the current accuracy of the drag data is insufficient to perform L/D analysis. It should be noted that accurate drag measurement on SD8020 airfoil at $Re = 4 \times 10^4$ is quite difficult to obtain, as neither publications [17, 43] by the airfoil designer included drag data at this low of a Reynolds number. Despite the inaccuracy of the drag data at low angle of attack, the data showed that a deployed passive flap has the potential to reduce drag past stall angle.

7.1.6 Full Model Results

Additional experiments were performed inside the wind tunnel on a Multiplex Merlin electric racer model over selected operational Reynolds number range of the aircraft. The aim of the study was to provide insight into the performance gain achievable when retrofitting existing aircraft design with the passive flap device. As the wing profile of the aircraft model is that of a thin airfoil with a slight camber, the stall pattern encountered should be similar to that of the SD8020 wing model; the model was fitted with a $c_f/c = 0.3$ flap at $x_f/c = 0.7$ and $x_f/c = 0.6$ position. The flap span was chosen to cover the portion of the wing with structural spar under the foam wing to prevent the deformation of the wing at high Reynolds number affecting the deployment of the flap. The experimental results are shown in Figure 7.12 below.

(a) $Re = 40000$

(b) $Re = 60000$

Figure 7.12: The lift curve of the aircraft mode with and without the passive flap device installed at $Re = 40,000$ and $Re = 60,000$.

It appears that the recirculation flow at $Re = 40,000$ is insufficient to overcome the weight of the flap, and no flap deployment was observed at this Reynolds number. On the other hand, the flap performance at $Re = 60,000$ lived up to the expectation and produced significant lift increase. It should be noted that flap-tipping was observed at 20 degree angle of attack as the airflow detached from the passive flap. The phenomenon is unlikely to be observed in actual flight as it occurred at an α significantly outside of the normal operating envelope of the aircraft, although flap angle limiter could be installed to address safety concern.

7.2 Simulation Results

The finite wing simulations aims to provide visualization of the three dimensional flow field around the wing, which would help us better understand the flow physics that led to the observations made in the finite wing experimental study. However, further testing of the transient solver had to be conducted to address the uncertainty regarding interactions between the `symmetryPlane` type boundary condition and the moving patch: if the mesh points on the symmetry patch remains stationary while the mesh points on, or connected to, the passive flap are moving, deformation of cells could occur which would affect the accuracy of the flow field, or cause the solver to crash in extreme case. Simulation testing for the behavior of the boundary layer setting was conducted with the 3D coarse mesh described in Section 5.2. The test case successfully produced results without error messages, and the resulting stream-trace plot is shown in Figure 7.13 below.

Figure 7.13: Ribbon stream trace for the 3D test case showing the flow field over the (a) upper and (b) lower surface of the wing.

The stream-trace in Figure 7.13 were integrated using Runge-Kutta 4/5 scheme, the trace lines were initiated⁵ at $y = 0.04$ and $y = -0.05$ for flow over the upper and lower surfaces, respectively; the ribbons were initialized with normal of 90° to the x-z plane. These stream-trace ribbons illustrated the 3D vortex structure at the wingtip (circled in red), where air flow swirled over the wing tip and cause the outer most portion of the streamlines to depress towards the upper surface of the airfoil; the ribbons at the upper wing surface in Figure 7.13(a) also showed the additional vortex generated by the edge of the flap (circled in blue), indicated by the changes in normal vector of the ribbons. Although the test case was unable to offer accurate prediction of flow separation, it showed that the selected boundary condition is feasible while giving us an idea of the 3D flow effects that might be observed.

7.2.1 Finite Wing Simulation Results

With the testing of the solver completed, simulations can be carried out for passive flap device on finite wing using the fine mesh described in Section 5.2. As the accuracy and applicability of the turbulence model and the FIM code has already been established in Section 6.2 for 2D airfoil simulations, the primary objective of this simulation was to evaluate the numerical solver as a practical research tool comparing to wind tunnel testing. To that end, the simulation output files gave us a wealth of data to analyze at the cost of high demand in computational resources. Regardless of the challenges faced, we were able to obtain velocity and pressure profiles throughout the span of the finite wing, as shown in Figure 7.14 & 7.15.

⁵ $(x, y) = (0, 0)$ is located at mid-chord of the airfoil geometry

(a) Velocity Profiles

(b) Pressure Profiles

Figure 7.14: The velocity and pressure profiles at $x = 0$ for the finite wing simulation with $\alpha = 14^\circ$ at $t = 1.1s$.

(a) Velocity Profiles

(b) Pressure Profiles

Figure 7.15: The velocity and pressure profiles at $x = 0.4$ for the finite wing simulation with $\alpha = 14^\circ$ at $t = 1.1s$.

The simulation was conducted using the super computer cluster in the Nanyang Technological University High Performance Computing Center. Sixteen computational cores across two nodes and up to 32GB of memory per nodes were utilized for the simulation of a flap configuration at one angle of attack; each of these simulations takes a better part of two weeks to perform 2s in simulated time, which is significantly longer than the time it took to perform the wind tunnel test with once the model was produced. The computational study, however, have the advantage of providing a more complete flow data comparing to the experimental study; it also allows us to post-process for certain flow data after the simulation was conducted, and revisit the flow results to measure variables not considered in the original study. The three-dimensional transient solution also allowed us to produce graphs visualizing the three dimensional flow effect on flow fields traveling over the upper and lower surfaces of the wing (Figure 7.16).

Figure 7.16: Stream traces initialized at $z = 2.0$ and $z = 1.0$ for finite wing at $\alpha = 14^\circ$ and $t = 1.1s$.

Figure 7.16 showed a larger displacement in the z -direction between the upper and lower stream ribbon in flow initiated at $z = 2.0$ than the one initiated at $z = 1.0$ (circled in blue); the observation shows that the flow feature that cause the displacement in stream traces is most likely initiated at the wingtip, but have little effect on flow closer to the root of the wing. By combining the observations from Figure 7.14, 7.15 and 7.16, we can reach the conclusion that the primary three dimensional flow effect influencing the performance of the passive flap is the wingtip vortex. As the lift enhancement from the passive flap is due to its position at the optimal flap angle, the non-uniform flow field due to wingtip vortex would cause the flap to reach an equilibrium β that is smaller than the optimal β , thus reducing the additional lift generated.

Mentioned as a motivation for the development of FIM code in Section 4.3.2, Large Eddy Simulation (LES) is generally considered to yield more accurate simulation results comparing to Unsteady Reynolds Average Navier-Stokes (URANS) simulation, especially at low Reynolds number like the current case; however, the LES simulation is also more expensive in terms of computational resources comparing to URANS simulation. In order to identify the method that would yield the best results with the given computational resources, the two turbulence modeling methods were tested using the $\alpha = 14^\circ$ model with $x_f/c = 0.8$, $c_f/c = 0.2$, and $b_f/b = 1.0$ flap configuration. The test case was chosen as we were certain that flow separation has occurred at that angle of attack, and the URANS flow solution is already available. The velocity and pressure profile at $z = 1.0$ and $z = 2.0$ from the LES and URANS simulations were compared in Figure 7.17 & 7.18.

(a) URANS, $z = 1$, $t = 1.0$

(b) LES, $z = 1$, $t = 1.0$

(c) URANS, $z = 1$, $t = 1.0$

(d) LES, $z = 1$, $t = 1.0$

(e) URANS, $z = 1$, $t = 1.25$

(f) LES, $z = 1$, $t = 1.25$

(g) URANS, $z = 1$, $t = 1.25$

(h) LES, $z = 1$, $t = 1.25$

Figure 7.17: Comparison of the velocity and pressure profiles at $z = 1.0$ for the finite wing simulation with $\alpha = 14^\circ$ using URANS and LES model.

(a) URANS, $z = 2$, $t = 1.0$

(b) LES, $z = 2$, $t = 1.0$

(c) URANS, $z = 2$, $t = 1.0$

(d) LES, $z = 2$, $t = 1.0$

(e) URANS, $z = 2$, $t = 1.25$

(f) LES, $z = 2$, $t = 1.25$

(g) URANS, $z = 2$, $t = 1.25$

(h) LES, $z = 2$, $t = 1.25$

Figure 7.18: Comparison of the velocity and pressure profiles at $z = 2.0$ for the finite wing simulation with $\alpha = 14^\circ$ using URANS and LES model.

The results from LES and URANS modeling showed very little difference when the velocity and pressure plots were compared, and the lift and drag data from the two simulations were nearly identical. This could be due to the fine boundary layer mesh removed the need for near wall treatment in the URANS simulation, and the turbulence level is relatively low given the low Reynolds number the simulations were conducted in. The low Reynolds number also means that sub-grid turbulence eddies in the recirculation zone and around the wing have very little impact on the overall flow solution. Judging from these results, it should be safe to say that the additional accuracy from the LES simulation does not pose enough advantage to warrant the additional cost over URANS simulation at the low Reynolds number of the current case.

7.3 Comparison and Discussion

Although the passive flap has previously been installed and tested on full-scale glider aircraft [12], no systematically collected data for passive flap on finite wing has been documented and studies until the wind tunnel experiments conducted in the current project. The experimental study gave us the data required to categorize the design variables for passive flap based on their effect on the performance of the device; it also confirmed our assumption on how c_f and x_f influence the flap performance based on the observations from 2D steady-state simulations, while providing additional data needed for the characterization of b_f in the design process such as the pressure contour presented in Figure 7.19.

Figure 7.19: The pressure contour plot at the upper and lower surfaces of the passive flap. The wing tip is on the left, and leading edge of the flap is at the minimal x value. The model orientation is $\alpha = 14^\circ$ and $t = 1.25s$.

As shown in the figure above, the 3D flow effect in the form of wing tip vortex influenced the pressure field acting across the flap, which causes the equilibrium β to deviate from its 2D equilibrium angle. The pressure contour indicated that the wing tip vortex induced an additional downward pressure, i.e. downwash, at the 10% of the wing span closest to the wing tip. This observation coincide with the experimental data that showed better passive flap performance for the $b_f/b = 0.8$ case comparing to the $b_f/b = 1.0$ case, as the shorter flap was able to deploy with minimal influence from the wing tip vortex.

It was thanks to the simulation setup that was used for boundary condition testing that we noticed the need to account for interference from cross flow when designing passive flap device with multiple span-wise segments. In the event that cross flow between different flap segments produces sufficient

pressure difference across the flap, individual flaps would deviates from the optimal flap angle, resulting in diminished, if not negative, lift enhancement. This phenomenon helps explained the markedly different results obtained in the two-segment flap configuration used in the current project and the data obtained in previous studies [11, 14].

The prospect of using numerical simulation as a primary research tool for the current project suffered greatly due to the amount of time required to obtain accurate simulation results, which is up to two weeks for the three-dimensional unsteady-RANS simulations; the time required to complete the LES simulation is even longer. Due to the wealth of flow data contained in the numerical solutions the simulation result turned out to be a good source of supplemental information for better understanding of experimental observations; however, the simulation time required with the hardware resource available for the current project made it an unattractive option comparing to wind tunnel experiments. For now, the numerical simulation could only be used for in depth analysis of specific cases for flow characteristics that could not be easily measured in the wind tunnel. Given the current simulation setup, it would be advisable to conduct c_f and x_f studies in two-dimensional transient cases, obtain initial b_f value with lift distribution study using steady-state simulation, then finalize the design process using 3D transient simulation.

Chapter 8

Conclusion

The current project started out with a simple objective: to employ the OpenFOAM solver to replicate the results from the passive flap experiments obtained in the water tunnel facility. As simulation results in general contain a lot more information than the readings from the force balance, a well calibrated simulation solver could shed more light on the flow interaction between the separated flow and the passive flap. The information, in this case lift data from passive flap usage, can then be used to create design criteria for the passive flap used by aircraft under the more complicated, real life, flight conditions. The project soon expanded to include experimental studies of employing the passive flap on finite-wing model, which we could find no prior studies containing data for, and the adaptation of the numerical solver to handle 3D simulation of finite wing with finite flap span.

It should be noted that there remains outstanding issues to be addressed regarding the numerical results, in particular regarding the near-wall mesh quality of the mesh used in the steady-state simulations. While the simu-

lated integral aerodynamic characteristic, such as lift, from both steady-state and transient solvers agree reasonably well with the experimental data, the computed flow details along the airfoil surface will require further validation.

8.1 Achievements

The present study was conducted in two parts: the experimental study conducted at the Nanyang Technological University wind tunnel facility, and the simulation study conducted with custom solver based on the OpenFOAM source code. We were able to gather valuable data through the experimental study that helped answer questions such as the effect of wingtip vortex on passive flap performance, the performance of passive flap on existing MAV-scale-glider airframe, and the performance of multi-segments passive flap design on straight, finite wing. Due to limitation of measuring instruments, we were only able to present the lift curve for the different passive flap configurations tested. The available data, however, enabled us to establish the effect of passive flap configurations, and Reynolds number on the lift generation of passive flap. When complemented with the simulation data, we were able to establish the design guideline for optimal configuration of passive flap high lift device for MAV with rectangular wing.

As an open source solver, the OpenFOAM source code possesses a steep learning curve for new users and programmers alike due to the lack of documentation and inconsistent programming styles; on the other hand, since the source code is freely available, we were able to modify and customize the solver to fit our specific need once a sound understanding of the code base

was achieved. The current study successfully developed an algorithm that combined various existing solvers and OpenFOAM libraries to construct the steady-state and transient fluid-induced-motion solvers for the simulation of passive flap device. The development process and the different turbulence modeling used was documented in Chapter 6. Currently, only comparisons of the calculated lift with experimental data have been carried out, but not for the flow structures.

By using data from the experimental study and supplementing it with simulation studies, we were able to determine the design guideline for variables that influence the effectiveness of the passive flap device in terms of lift production. Data gathered in this study suggested that the determination of the flap span b_f could be achieved by studying the lift distribution along the wing span; the flap position x_f and flap length c_f depends on the wing profile: x_f depends on the location of initial flow separation, i.e. the curvature and thickness of the front section of the wing profile, while c_f depends on the curvature of the airfoil after x_f . Based on the aforementioned observations, we were able to use the experimental results supported by simulation data for the determination of the previously mentioned variables.

8.2 Future Work

The current project investigated the performance of the “Pop-Up Feather” type passive lift enhancement device based on its three main design variables: the chord-wise position x_f , the span-wise dimension b_f , and the length of the flap c_f . The implementation of the passive flap as a practical lift enhance-

ment device, however, requires the investigation of additional variables such as: the weight and material of the flap; the effect of the flap on lift distribution and the structural integrity of the flap; and the effect of cross wind on flap. Further experiments are also required to explore the usage of flap angle limiter as well as mechanism that would lock down the passive flap when its deployment is not desirable.

While the simulation results from the current 2D solver agree well with the experimental data in terms of integral quantities, such as lift, further validation is advisable if more detailed experimental data, such as PIV measurements, are available. Furthermore, more validation of the 3D solver is desirable, as the high computational cost of these simulations were beyond the scope of this thesis.

Future development for the transient solver should concentrate on adapting the solver so that it is compatible with the latest distribution of OpenFOAM, as the source code has gone through extensive development and reorganization during the transition from version 1.x to 2.x. The newer version of OpenFOAM source code also includes the Adaptive Mesh Interface (AMI) type boundary condition, which would make it possible to simulate passive flap with multiple segments without creating highly distorted mesh between the different segments. Validation and verification study should also be done to establish the accuracy of drag prediction and unsteady flow features to aid future studies, such as the dynamic pitching of wing with flap and changes in lift over drag ratio.

Bibliography

- [1] Weller, H., Tabor, G., Jasak, H., and Fureby, C., “A Tensorial Approach to Computational Continuum Mechanics using Object-Oriented Techniques,” *Computer in Physics*, Vol. 12, No. 6, 1998, pp. 620–631.
- [2] Editors, “More About Balloons,” *Scientific American*, Vol. 4, No. 26, 1849, pp. 205.
- [3] *DoD Dictionary of Military Terms*, Department of Defence, The United States, 2011.
- [4] Blake, B., *Jane’s Weapon System 1987-1988*, Jane’s Information Group, 1988.
- [5] Munson, K., *World Unmanned Aircraft*, Jane’s Information Group, 1988.
- [6] AeroVironment Inc., “Raven UAS Launch,” http://www.avinc.com/img/media_gallery/UAS_Raven_Field_Launch.jpg, retrieved Jan 17th, 2012.

- [7] U.S. Air Force, “Predator soars to record numbers of sorties,” <http://www.af.mil/news/story.asp?id=123063918>, August 2007, retrieved Jan 17th, 2012.
- [8] Furlong, G. and Fitzpatric, J., “Effects of Mach Number and Reynolds Number on the Maximum Lift Coefficient of a Wing of NACA 230-Series Airfoil Section,” Tech. Rep. 1299, NACA, 1947.
- [9] Schatz, M., Knacke, T., Thiele, F., Meyer, R., Hage, W., and Bechert, D., “Separation Control by Self-Activated Movable Flaps,” *42th AIAA Aerospace Sciences Meeting & Exhibit*, Reno, NV, 2004, AIAA 2004-1243.
- [10] Liebe, W., “Der Auftrieb am Tragflügel: Entstehung und Zusammenbruch,” *Aerokurier*, Vol. 12, 1979, pp. 1520–1523.
- [11] Patone, G. and Müller, W., “Aeroflexible Oberflächenklappen als ”Rückstrombremsen” nach dem Vorbild der Deckfedern des Vogel- flügels,” Tech. Rep. TR-96-05, TU Berlin, 1996.
- [12] Bechert, D., Bruse, M., Meyer, R., and Hage, W., “Biological Surfaces and Their Technological Application – Laboratory and Flight Experiments on Drag Reduction and Separation Control,” *AIAA*, No. 97-1960, 1997.
- [13] Bramesfeld, G. and Maughmer, M., “Experimental Investigation of Self-Acutating, Upper-Surface, High-Lift-Enhancing Effectors,” *Journal of Aircraft*, Vol. 39, No. 1, January-February 2002, pp. 120–124.

- [14] Kernstine, K., Moore, C., Cutler, A., and Mittal, R., “Initial Characterization of Self-Activated Movable Flaps, ”Pop-Up Feathers”,” *46th AIAA Aerospace Sciences Meeting & Exhibit*, Reno, NV, 2008, AIAA 2008-369.
- [15] Schlüter, J., “Lift Enhancement at Low Reynolds Numbers using Pop-Up Feathers,” *39th AIAA Fluid Dynamics Conference*, San Antonio, TX, 2009, AIAA 2009-4195.
- [16] Johnston, J., Gopalarathnam, A., and Edwards Jr., J., “Experimental Investigation of Bio-Inspired High Lift Effectors on a 2-D Airofil,” *29th AIAA Applied Aerodynamics Conference*, Honolulu, Hawaii, 2011, AIAA 2011-3791.
- [17] Selig, M., Guglielmo, J., Broern, A., and Giguere, P., “Experiments on Airfoil at Low Reynolds Numbers,” *34th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, 1996, AIAA 96-0062.
- [18] Barlow, J., Rae Jr., W., and Pope, A., *Low-Speed Wind Tunnel Testing*, Wiley-Interscience, 3rd ed., 1999.
- [19] *MULTIPLEX Merlin Build Instruction*, Multiplex Modellsport GmbH & Co.KG, 2009.
- [20] Franke, J., Sturm, M., and Kalmbach, C., “Validation of OpenFOAM1.6.x with the German VDI guideline for obstacle resolving micro-scale models,” *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 104-106, 2012, pp. 350–359.

- [21] Yabili, S., Smith, M., and Dimitriadis, G., “Unsteady Navier-Stokes Simulation of Low-Reynolds Stall Flutter,” *50th AIAA Aerospace Meeting*, Nashville, Tennessee, 2012.
- [22] Ghisu, T., Jarrett, J., and Parks, G., “Robust Design Optimization of Airfoils with Respect to Ice Accretion,” *Journal of Aircraft*, , No. 1, 2011, pp. 287–304.
- [23] Doolan, C., “Flat-Plate Interaction with the Near Wake of a Square Cylinder,” *AIAA Journal*, , No. 2, 2009, pp. 475–479.
- [24] ICON CFD, “Volkswagen Group Adopts New Computational Fluids Dynamics (CFD) Process Developed with ICON,” October 07, 2009, Press Release.
- [25] Patankar, S., *Numerical Heat Transfer and Fluid Flow*, Taylor & Francis, 1980.
- [26] Issa, R., “Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting,” *Journal of Computational Physics*, Vol. 62, 1985, pp. 40–65.
- [27] Spalart, P. and Allmaras, S., “A One-Equation Turbulence Model for Aerodynamic Flows,” *La Recherche A’rospatiale*, , No. 1, 1994, pp. 5–21.
- [28] Ashford, G., *An Unstructured Grid Generation and Adaptive Solution Technique for High Reynolds Number Compressible Flows*, Ph.D. thesis, University of Michigan, 1996.

- [29] Menter, F., “Zonal Two Equation k - ω Turbulence Models for Aerodynamic Flows,” *24th Fluid Dynamics Conference*, Orlando, FL, 1993, AIAA 93-2906.
- [30] Menter, F., “Two-Equations Eddy-Viscosity Turbulence Models for Engineering Applications,” *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.
- [31] Menter, F. and Esch, T., “Elements of Industrial heat Transfer Predictions,” *16th Brazilian Congress of Mechanical Engineering*, Uberlandia, Brazil, 2001.
- [32] Smagorinsky, J., “General Circulation Experiments with Primitive Equations: I. The Basic Experiments,” *Monthly Weather Review*, Vol. 91, No. 3, 1963, pp. 99–164.
- [33] M. Germano, U. Piomelli, P. M. and Cabot, W., “A Dynamic Subgrid-Scale Eddy Viscosity Model,” *Physics of Fluids*, Vol. 3, No. 7, 1991, pp. 1760–1765.
- [34] Lilly, D., “A Proposed Modification of the Germano Subgrid-scale Closure Method,” *Physics of Fluids*, Vol. 4, No. 3, 1992, pp. 633–635.
- [35] Lesieur, M. and Métais, O., “New Trends In Large-Eddy Simulations of Turbulence,” *Annual Review of Fluid Dynamics*, Vol. 28, 1996, pp. 45–82.

- [36] Meneveau, C. and Kat, J., “New Trends In Large-Eddie Simulations of Turbulence,” *Annual Review of Fluid Dynamics*, Vol. 32, 2000, pp. 1–32.
- [37] *OpenFOAM User Guide*, OpenCFD Ltd., 2010.
- [38] Kassiotis, C., “Which strategy to move the mesh in the Computational Fluid Dynamic code OpenFOAM.” *Report for the PhD course in OpenFOAM at Chalmers TH*, Göteborg, Sverige, 2008.
- [39] Jasak., H. and Tukovic, Z., “Automatic Mesh Motion for the Unstructured Finite Volume Method,” *Transactions of FAMENA*, Vol. 30, No. 2, 2007.
- [40] Ekedahl, E., “6-DOF VOF-solver without Damping in OpenFOAM,” *Report for the PhD course in OpenFOAM at Chalmers TH*, Göteborg, Sverige, 2009.
- [41] Schatz, M., *Numerische Simulation der Beeinflussung instationärer Strömungsablösung durch frei bewegliche Rückstrom-klappen auf Tragflügeln*, Ph.D. thesis, Technische Universität Berlin, 2003.
- [42] Wang, C. and Schluter, J., “Stall control with feathers: Self-activated flaps on finite wings at low Reynolds numbers,” *Comptes Rendus Mécanique*, Vol. 340, No. 1-2, 2012.
- [43] Selig, M., Donovan, J., and Fraser, D., *Airfoils at Low Speeds*, Vol. 8, SoarTech Publications, Virginia Beach, VA, 1989.