Fluid-Structure Interaction Simulations of a Flapping Wing Micro Air Vehicle

Alex W. Byrd
Wright State University

Follow this and additional works at: https://corescholar.libraries.wright.edu/etd_all

Part of the Mechanical Engineering Commons

Repository Citation
https://corescholar.libraries.wright.edu/etd_all/1212

This Thesis is brought to you for free and open access by the Theses and Dissertations at CORE Scholar. It has been accepted for inclusion in Browse all Theses and Dissertations by an authorized administrator of CORE Scholar. For more information, please contact library-corescholar@wright.edu.
FLUID-STRUCTURE INTERACTION SIMULATIONS OF A FLAPPING WING MICRO AIR VEHICLE

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Engineering

By

ALEXANDER WAYNE BYRD
B.S., Wright State University, 2012

2014
Wright State University

George Huang, Ph.D.
Thesis Director

Committee on Final Examination

Philip Beran, Ph.D.

Joseph Shang, Ph.D.

George Huang, Ph.D.
Chair
Department of Mechanical and Materials Engineering
College of Engineering and Computer Science

Robert E. W. Fyffe, Ph.D.
Vice President for Research and Dean of the Graduate School
ABSTRACT

Byrd, Alex Wayne. M.S., Egr., Wright State University, 2014.
Fluid-Structure Interaction Simulations of a Flapping Wing Micro Air Vehicle

Interest in micro air vehicles (MAVs) for reconnaissance and surveillance has grown steadily in the last decade. Prototypes are being developed and built with a variety of capabilities, such as the ability to hover and glide. However, the design of these vehicles is hindered by the lack of understanding of the underlying physics; therefore, the design process for MAVs has relied mostly on trial-and-error based production. Fluid-Structure Interaction (FSI) techniques can be used to improve upon the results found in traditional computational fluid dynamics (CFD) simulations. In this thesis, a verification of FSI is first completed, followed by FSI MAV simulations looking at different prescribed amplitudes and flapping frequencies. Finally, a qualitative comparison is made to high speed footage of an MAV. While the results show there are still model improvements that can be made, this thesis hopes to be a stepping stone for future analyses for FSI MAV simulations.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>2. BACKGROUND</td>
<td>7</td>
</tr>
<tr>
<td>3. VERIFICATION OF FSI</td>
<td>17</td>
</tr>
<tr>
<td>4. MAV CASE SETUP</td>
<td>32</td>
</tr>
<tr>
<td>5. MAV CASE INTRODUCTION</td>
<td>32</td>
</tr>
<tr>
<td>WING GEOMETRY</td>
<td>32</td>
</tr>
<tr>
<td>Uniform Modulus Wing</td>
<td>32</td>
</tr>
<tr>
<td>ABAQUS (FEA) SETUP</td>
<td>18</td>
</tr>
<tr>
<td>Material and Section Properties</td>
<td>18</td>
</tr>
<tr>
<td>Boundary Conditions</td>
<td>19</td>
</tr>
<tr>
<td>Meshing and Element Choices</td>
<td>20</td>
</tr>
<tr>
<td>Other ABAQUS Conditions</td>
<td>22</td>
</tr>
<tr>
<td>SC/TETRA (CFD) SETUP</td>
<td>22</td>
</tr>
<tr>
<td>Volume Region</td>
<td>22</td>
</tr>
<tr>
<td>Meshing</td>
<td>23</td>
</tr>
<tr>
<td>Boundary Conditions</td>
<td>25</td>
</tr>
<tr>
<td>Other Conditions</td>
<td>28</td>
</tr>
<tr>
<td>VERIFICATION RESULTS</td>
<td>29</td>
</tr>
<tr>
<td>4. MAV CASE SETUP</td>
<td>32</td>
</tr>
<tr>
<td>MAV CASE INTRODUCTION</td>
<td>32</td>
</tr>
<tr>
<td>WING GEOMETRY</td>
<td>32</td>
</tr>
<tr>
<td>Uniform Modulus Wing</td>
<td>32</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>1.1</td>
<td>Insect wing of a cicada broken into four regions. (Credit to [14])</td>
</tr>
<tr>
<td>2.1</td>
<td>Diagram showing iterative FSI methodology.</td>
</tr>
<tr>
<td>3.1</td>
<td>Verification case geometry.</td>
</tr>
<tr>
<td>3.2</td>
<td>Isometric view of encastre boundary condition.</td>
</tr>
<tr>
<td>3.3</td>
<td>Isometric view of z-symmetry boundary condition.</td>
</tr>
<tr>
<td>3.4</td>
<td>View of mesh in x-z plane showing the full span of the brick elements along the z direction.</td>
</tr>
<tr>
<td>3.5</td>
<td>Isometric view of mesh showing the layers of elements in x and y directions.</td>
</tr>
<tr>
<td>3.6</td>
<td>Isometric view of CFD master volume.</td>
</tr>
<tr>
<td>3.7</td>
<td>Octant setup for mesh showing three regions of refinement.</td>
</tr>
<tr>
<td>3.8</td>
<td>View of different size of prism layers along top wall and cylinder wall.</td>
</tr>
<tr>
<td>3.9</td>
<td>Overall CFD mesh including octant sizing and prism layers.</td>
</tr>
<tr>
<td>3.10</td>
<td>Velocity profile at inlet of master volume.</td>
</tr>
<tr>
<td>3.11</td>
<td>Summary of boundary conditions for CFD side of verification case.</td>
</tr>
<tr>
<td>3.12</td>
<td>Y-displacement data from four different cases.</td>
</tr>
<tr>
<td>3.13</td>
<td>Results of y-displacement of point A at a 50 microsecond timestep.</td>
</tr>
<tr>
<td>4.1</td>
<td>Rectangular wing developed for comparison purposes.</td>
</tr>
<tr>
<td>4.2</td>
<td>Experimental wing with approximate dimensions in milimeters.</td>
</tr>
<tr>
<td>4.3</td>
<td>The two wing sections corresponding to the material properties in table 4.1.</td>
</tr>
<tr>
<td>4.4</td>
<td>Rigid nodes on the holder of the wing (nodes are shown from both sides).</td>
</tr>
<tr>
<td>4.5</td>
<td>Surfaces of the holder and wing that are tied together.</td>
</tr>
</tbody>
</table>
Figure 4.6: Region where the carbon fiber branches are tied to the film.......................... 37
Figure 4.7: Four-bar simulation data calculated in Matlab........................................... 39
Figure 4.8: Mesh of the holder with brick elements..................................................... 40
Figure 4.9: Mesh of the experimental wing with both brick and shell elements.......... 41
Figure 4.10: Hemispherical volume used for master region in simulations............... 42
Figure 4.11: y-z plane view of master region with tail included.............................. 43
Figure 4.12: Isometric view of rotating region......................................................... 44
Figure 4.13: Cross-sectional view of rotate region, noting 2 prism layers near wall of
wing.................................................................................................................. 45
Figure 4.14: x-z plane of mesh with boundary conditions labeled............................ 47
Figure 4.15: Cross-sectional view of the wing inside the overset mesh, noting how the
mesh gets coarser farther away from the rotating region..................................... 48
Figure 5.1: Lift history of 5 Hz rectangular wing model.............................................. 51
Figure 5.2: Pressure contours at starting position (0.2 seconds)................................. 52
Figure 5.3: Pressure contours at 0.24 seconds............................................................ 52
Figure 5.4: Pressure contours at 0.28 seconds............................................................. 52
Figure 5.5: Pressure contours at 0.32 seconds............................................................. 53
Figure 5.6: Pressure contours at 0.36 seconds............................................................. 53
Figure 5.7: Lift history for 15 Hz of rectangular wing................................................. 54
Figure 5.8: Lift history for half a second of 5 and 15 Hz of the rectangular wing........ 55
Figure 5.9: Lift history of experimental wing model at 15 Hz.................................... 56
Figure 5.10: Pressure contours on wing mid down-stroke at 0.016 seconds............. 57
Figure 5.11: Pressure contours on wing at bottom of cycle at 0.032 seconds............ 57
Figure 5.12: Pressure contours on wing mid up-stroke at 0.048 seconds. ....................... 58
Figure 5.13: Pressure contours on wing near the start of its cycle at 0.074 seconds. .... 58
Figure 5.14: Lift history of sinusoidal, 120º amplitude case. ................................. 59
Figure 5.15: Lift history of four-bar motion, 120º amplitude case. .......................... 60
Figure 5.16: Side by side comparison of four bar and sinusoidal motion. ................. 61
Figure 5.17: High speed camera footage and simulation at beginning of cycle. ........ 62
Figure 5.18: High speed camera footage and simulation at upper end of stroke. ....... 62
Figure 5.19: High speed camera footage and simulation at mid up-stroke. ............... 62
Figure 5.20: High speed camera footage and simulation at mid down-stroke .......... 63
Figure 5.21: Deformation of wing with reduced modulus in the branches. ............... 64
<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Material properties of the flexible attachment</td>
<td>18</td>
</tr>
<tr>
<td>3.2</td>
<td>Element sizing for coarse and refined meshes</td>
<td>24</td>
</tr>
<tr>
<td>3.3</td>
<td>Basic CFD Settings</td>
<td>28</td>
</tr>
<tr>
<td>3.4</td>
<td>Validation Case Setup</td>
<td>29</td>
</tr>
<tr>
<td>3.5</td>
<td>Max displacements for various cases</td>
<td>31</td>
</tr>
<tr>
<td>4.1</td>
<td>Material properties for the wings</td>
<td>34</td>
</tr>
<tr>
<td>4.2</td>
<td>Boundary condition values</td>
<td>37</td>
</tr>
<tr>
<td>4.3</td>
<td>Element sizes for master and rotating region</td>
<td>45</td>
</tr>
<tr>
<td>4.4</td>
<td>Basic CFD settings</td>
<td>48</td>
</tr>
<tr>
<td>4.5</td>
<td>Fluid properties of air</td>
<td>49</td>
</tr>
</tbody>
</table>
NOMENCLATURE

\( u, v, w \)  x, y, z direction velocity [m/s]
\( g \)  Acceleration due to gravity [m/s\(^2\)]
\( p \)  Pressure [Pa]
\( \mu \)  Dynamic viscosity [kg/(m-s)]
\( \rho \)  Density [kg/m\(^3\)]
\( \nabla \)  Del operator
\( \vec{v} \)  Velocity vector [m/s]
\( U_i \)  Mean velocity in tensor notation [m/s]
\( S_{ji} \)  Mean strain-rate tensor [1/s]
\( \bar{u}_i' u_i' \)  Temporal average of fluctuating velocities [m/s]
\( \nu \)  Kinematic molecular viscosity [m\(^2\)/s]
\( \nu_T \)  Kinematic eddy viscosity [m\(^2\)/s]
\( k \)  Kinetic energy of turbulent fluctuations per unit mass [m\(^2\)/s\(^2\)]
\( \varepsilon \)  Dissipation per unit mass [m\(^2\)/s\(^3\)]
\( \tau_{ij} \)  Specific Reynolds stress tensor [N/m\(^2\)]
\( C_{\mu,\varepsilon_1,\varepsilon_2}, \sigma_{k,\varepsilon} \)  Closure coefficients for k- \( \varepsilon \) turbulence model
\( \lambda \)  Stretch ratio
\( \mathbf{x} \)  Coordinates in space after period of time t [m]
\( \mathbf{X} \)  Coordinates initially in space [m]
\( \varepsilon_s \)  Strain
\( \mathbf{n} \)  Unit vector corresponding to Eigenvector \( \mathbf{N} \)
\( \mathbf{V} \)  Left stretch matrix
\( c \)  Undetermined parameter
\( \alpha, \beta \)  Time dependent parameters
\( t \)  Time [s]
\( \mathbf{M} \)  Mass matrix [kg]
\( \mathbf{C} \)  Damping matrix [N-s/m]
\( \mathbf{K} \)  Stiffness matrix [N/m]
\( \mathbf{a} \)  Displacement matrix [m]
\( \mathbf{f} \)  Force matrix [N]

\( Q \)  Prescribed displacement matrix [m]

\( d_{k+1}^{n+1} \)  Amount of mesh movement at (k)

\( \tilde{d}_{k+1}^{n+1} \)  Received displacement from structural analysis at (k+1)

\( d_{k}^{n+1} \)  Amount of mesh movement at (k+1)

\( \omega \)  Under-relaxation parameter
ACKNOWLEDGEMENTS

I first would like to thank my advisor, Dr. George Huang, for his guidance and knowledge through the length of this project. Thanks to his resources, I was funded for both classwork and this research through the Wright State Graduate Council. His experience in the field of CFD proved invaluable when research hit the expected bumps along the way. I am very grateful that this research has led me to my current job. Dr. Huang’s former student, Sunil, has been an excellent help to me throughout this project. His work on the validation and guidance for both CFD and Fluid-Structure Interaction techniques helped me get past some of the most frustrating portions of my research.

I would also like to thank Dr. Nathan Klingbeil. His ability to inspire students in and out of the classroom has always been something I have greatly admired. I enjoyed the chance to work on his teaching team and assist freshman with understanding important math concepts for engineering. He encouraged me to pursue my aspirations in this field, which led me to work with Dr. Huang.

A number of family and friends have been a great help on this project. My girlfriend Dalila has been very supportive through the entire project, as there were many late nights I would arrive to an excellent home-cooked meal and some much needed time to relax. My best friend throughout college, Michael, has been an excellent source of knowledge through his experiences with computational modeling and CFD. My friend Jayme helped greatly by getting my foot in the door at a local company, for which I am excited to continue work with numerous types of CFD simulations.

Last but not least, I would like to thank my father for his great help on this project. I am truly thankful to be able to openly discuss with him many engineering
questions and concerns. On numerous occasions I have been able to rely on his many
years of research and expertise in the thermodynamics field for difficult problems. His
help with proofreading this thesis and encouragement to keep working on it until it is
finished has ensured I see this project to its completion. His advice to always “do more
than the minimum required” has paved the road for me to strive for excellence in my
career, and my life in general.
1. INTRODUCTION

Micro air vehicles (MAVs) have been in heavy development since the early 1990’s [1]. Being much smaller than previously designed aircraft, they run into unique aerodynamic challenges [2]. While their primary initial interest was for military reconnaissance, a number of other possibilities have cropped up over time. For example, emergency first-responders could use MAVs for everything from fires to natural disasters to search for trapped people where it might not be safe for them to go in themselves. Another use is surveillance in areas that might not be feasible to send in people, such as over difficult terrain. All applications will need durability to survive various conditions in aforementioned environments.

For the most part, MAVs are separated into three different types: fixed wing, rotary wing, and flapping wing derivatives. While fixed and rotary wing options have advantages, they are generally much larger than flapping wing MAV’s. The design advantage of a flapping wing MAV is that it has existed in nature for thousands of years through insects and birds, such as dragonflies and hummingbirds. That being said, mimicking nature in this case has proven not to be an easy task. Weight is very critical, and more complex designs often become heavier as well.

A number of research studies have modeled the flapping wing MAV as a rigid plate and done further analysis based on this assumption. Going back a few decades, a study was done looking at fluid-dynamic efficiency with rigid and flexible plates. The study showed a general optimum shape for a wing, and demonstrated the important effect a wing’s shape had on flight performance [3]. While this is not actually the case in
nature, it has still been used to yield good results in minimizing power consumption of the MAV [4]. A rigid plate was also used with a torsional spring model and investigated passive deflection of the plate [5].

How insects are able to generate lift is an important concept towards replicating it for the development of MAVs. Wootton’s research looked at the unique parts of how an insect flies: the flapping wings and the high deformations. His analogy of insect wings to sails on boats emphasized the important similarity of a flexible membrane supported by rigid spars [6].

Numerous research studies have looked into flight of the *manduca sexta* (hawkmoth). Experiments with high speed footage have shown a stroke amplitude close to 120 degrees, and reasonable symmetry between left and right wings during forward and hovering flight. Additionally, dramatic bending and twisting of the wings was observed during slow and hovering flight [7]. One computational model of flight of the hawkmoth was created using a simplified wing shape and specifying its shape changes in flight ahead of time [8]. While this can handle examining known wings with documented flight shapes at various times, it has limitations when looking into varying wing shapes with unknown flight characteristics. One experiment looked at the structural response of the hawkmoth, recording the first three modes of the wing to be 59, 75, and 95 Hz [33]. These modes were found using a scanning laser vibrometer, and were shown to be about 26% higher when tested in vacuum rather than air. Experiments done by DeLeón and Palazotto showed a hawkmoth wing behaved significantly different in air versus vacuum, suggesting nature was able to take advantage of aerodynamic forces difficult to replicate with an engineered wing [34]. Later, Hollenbock and Palazotto summarize recent
experimental and computational models, including nanoindentation methods for characterizing material properties [32].

Another study was done investigating the motion of the wings of insects during flight. Flight characteristics of a ladybug showed the wings followed a more “figure-8” like motion as opposed to straight up and down [9]. The ladybug has wings specifically designed to deflect at the end of the stroke, along with pitch change due to the clap-and-fling effect. This effect, where each of the wings go through a full 180 degree rotation, resulted in increased lift without any “active” controls. This simply means the wings passively adjust due to aerodynamic forces into favorable pitch changes and wing spar deformations [9].

Some relevant studies for determining wing shape in flight investigated the comparison of fluid-dynamic and inertial-elastic forces on the wing. Combes and Daniel compared wing flight of MAVs in both standard air and a helium environment. This allowed for an investigation of the effect of aerodynamic forces, as these would be significantly reduced in the much lower air density of a mostly-helium fluid. The results showed that the wing patterns during flight changed only minimally, suggesting that aerodynamic forces play only a small role compared to wing inertia [10]. It further suggested that air tends to cause more of a fluid damping rather than significant aerodynamic forces, which could allow for simpler modeling in future cases.

Wilkin and Williams performed experiments with a camera and strain-gauge probe to measure instantaneous vertical and horizontal forces on a sphingid moth. Their results suggested that inertial forces are a larger consideration during flight than aerodynamic forces [11]. However, a computational fluid dynamics (CFD) study looking
at lift and power requirements of a hovering *drosophila virilis* (fruit fly) concluded the opposite; that aerodynamic forces tended to be larger than inertial forces in flight [12]. While different species, the similar wings and flight patterns with conflicting conclusions highlight the need for further research to better quantify the results. Ennos’ study of *diptera* (flies) showed that wing inertia alone could cause the large twisting observed in wings when transferring from downstroke to upstroke [13].

Work has also been done that studied insect wing structure, with one common method to define various regions that is referred to as the Comstock-Needham System. The system breaks down the veins in the wings into six different categories based on location. A slightly different breakdown was done by Dawson et al. who broke the wings down into four regions as seen in figure 1.1 below:

![Figure 1.1: Insect wing of a cicada broken into four regions. (Credit to [14])](image)

By dividing the wing into those specified regions, it was determined which areas carried the highest load and would require higher strength. The research resulted in a design that had a higher stiffness in regions 1 and 2, while tapering off to a lower stiffness in regions 3 and 4 [14]. This allows the wing to twist and adjust angle of attack for lift.

Numerous experiments have investigated quantifying material properties of insect wings. Wootton noted that while general principles have been developed, there are major kinematic differences between morphologically similar insects [15]. These kinematic
differences lead to a need for very accurate modeling of wing geometry and wing flight for accurate results. Steppan measured bending stiffness in dried butterfly wings using cantilever loading. The study found a correlation between flexural stiffness and wing area, which could be useful when designing wings of various sizes [16]. Looking at a variety of insects, Combes and Daniel showed results for the flexural stiffness of spanwise and chordwise directions scaled with the cube of wing span and the square of chord length [17]. This research lays out the importance of the geometry of the wing veins to capture the anisotropic behavior during flight. Geometry must be modeled realistically to represent the varying stiffness along the span and chord of the wing. Further experiments by Combes and Daniel analyzed local stiffness along varying points of the wing. Using point loads on the wings and measuring displacements, they found an exponential decline for the stiffness model accurately predicted displacements in chordwise and spanwise directions [18].

Research on the morphology of dragonfly wings notes the importance of their pleated arrangement of the veins. Bending experiments found they stiffen the wing from spanwise bending moments, and are further helped from the membrane [19]. One of the struggles for computational modeling noted by Newman and Wootton are the unknown properties of the membrane itself.

A large part of the research for these vehicles is done experimentally, often in wind tunnels or load testing cells [9,14]. This thesis follows a different approach, expanding on models made using CFD. The particular nature of micro vehicle wings and the medium through which they are used (air) present some problems with traditional CFD models. While the model will certainly run (and has successfully), a certain amount
of accuracy is lost if particular care is not taken to describe the interaction between the wings and their environment. Prescribing motion using CFD alone will not take into account deformation of the wings due to the effect of the air. It is for this reason this thesis looks into a coupled structural-CFD analysis, known as Fluid-Structure Interaction (FSI).
2. BACKGROUND

Computer technology advancements in the last fifty years has allowed for the approach of CFD to flourish in both research and industry. CFD has become very useful for a wide variety of problems. First, many classical problems that are solved analytically cannot be applied to complex problems in today’s world. Due to complex geometries, material properties, boundary conditions and other factors, many of the assumptions in analytical solutions are not valid and fail to give an accurate solution. Additionally, while very useful, many experimental approaches are costly and/or time consuming. While experimental tests are still necessary to validate many problems solved numerically through CFD, one can use the numerical approach to help better design experimental tests. It can also help foresee some unexpected problems or issues that might require a test to be designed differently.

CFD takes a fluid domain and discretizes that region into nodes that form volumes. The SC/Tetra solver used for this analysis uses an unstructured finite-volume mesh to form tetrahedrons throughout the domain. The code uses a second-order accurate MUSCL scheme to solve for spatial terms, and an implicit first-order scheme for time derivative terms [25]. Tetrahedrons are able to approximate many different geometry shapes, making them an excellent choice (compared to brick hexahedral elements, for example) to handle complex geometries. Within the volumes, a number of equations are solved at the nodes of the tetrahedron which govern fluid flow. The governing equations which are solved for these simulations are the momentum conservation equations (Navier-Stokes), continuity equation, and energy equations.
Additional relations describe the thermodynamic equation of state and the turbulence model.

The Navier-Stokes equations for a Newtonian fluid are derived in a number of texts with a rather long set of differential equations. A few simplifications can be made for this application of MAV’s, such as incompressible flow and constant density. This is due to these simulations looking at air at a relatively constant temperature and low velocity, which keeps those aforementioned assumptions valid. The resulting equations in 3-D are listed below:

\[
\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho g_x - \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \tag{1}
\]

\[
\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \rho g_y - \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \tag{2}
\]

\[
\rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \rho g_z - \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \tag{3}
\]

These equations have been studied quite in depth, with a full derivation shown in [20]. These partial differential equations are used in combination with the continuity equation to solve for the variables \(u, v, w, \) and \(p\). The continuity equation, reduced for incompressible flow, is shown below:

\[
\nabla \cdot \vec{v} = 0 \tag{4}
\]

Again, this coupled with equations 1-3, is used to solve for velocity and pressure values for numerous types of flows [21].

One additional important component that must be considered for CFD is the turbulence model. Turbulence is a complicated issue as it can be best described as random, chaotic behavior of a fluid. However, there are a number of ways to model turbulence. Turbulence is looked at from a statistical approach, typically using a time
averaging technique. Generally, for incompressible, constant-property flow the time averaging on the Navier-Stokes equation results in the following equation:

$$\rho \frac{\partial u_i}{\partial t} + \rho U_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(2\mu S_{ij} - \rho \overline{u'_i u'_j})$$  \hspace{1cm} (5)

The fundamental issue of turbulence requires a way to prescribe the final term of equation 5. A more complete derivation of this can be found in [22]. A popular method used by this software is the two-equation standard k-ε model. This model has been shown to predict vortex shedding in highly separated, unsteady flows similar to that seen in MAV flight [31]. To avoid going into great detail about the formulation of this model, this thesis will just note the resulting equations that it produces to solve for kinematic eddy viscosity, turbulence kinetic energy, and dissipation rate:

$$\nu_T = C_\mu k^2/\varepsilon$$  \hspace{1cm} (6)

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (7)

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial \varepsilon}{\partial x_j} \right]$$  \hspace{1cm} (8)

Where the closer coefficients, \(C_{\varepsilon 1}\) and \(C_{\varepsilon 2}\), are explained in [22].

Finite Element Analysis (FEA) techniques are used to model the MAV structure. Structural analysis will generally use different element types compared to CFD, as well as solving different governing equations. For these simulations, both brick and shell elements were used. Brick elements (also referred to as hex elements) were used in both the validation and MAV simulations to model the flexible attachment (validation) and solid holder and branches of the MAV. These elements are simply a rectangular prism, with the standard version having eight nodes, one at each of the corners. Linear interpolation is used between the nodes, and is why proper mesh refinement is critical.
For increased accuracy, a quadratic version of this element also exists, which puts a node at the midsection between all of the original eight nodes. This results in a twenty node brick, and can be used when more accurate (though more time-consuming) results are critical. Shell elements were used to model the film attaching the branches of the wing together for the MAV simulations. The type chosen, referred to as S3 and S4 by ABAQUS, is a general-purpose shell element that has finite membrane strains. These elements form a triangle or quadrilateral depending on number of nodes to conform to the geometry, which is automatically determined by the software package. These elements work well in modeling the membrane of the MAV due to the fact that one direction (i.e., the thickness) of the film is negligible and can be ignored by the simulation. A number of complex equations are defined for each specific set of elements when solving for field equations, and can be found here [23].

As a general overview of FEA, the constitutive relation of stress and strain must be solved at all the nodes of the elements. ABAQUS solves for strain by first defining a stretch ratio as shown below:

\[ \lambda = \sqrt{\frac{dx^T \cdot dx}{dX^T \cdot dX}} \]  

(9)

It then considers strain to be a function of the defined stretch ratio. It should be noted that based on this formulation a stretch ratio of one is just rigid movement only [23]. Next, the strain can be defined in three dimensions as follows:

\[ \varepsilon_z = f(V) \]  

(10)

Where:
And is referred to as the “left stretch” matrix. While a lot of math is involved, it can then be shown for the change of strain equation [23]:

$$\Delta \varepsilon_s = \ln(\Delta \mathbf{V})$$

From this change of strain, one can calculate the stress at each of the nodes over time.

There are a number of ways to move forward in time, and a popular one for FEA is shown below:

$$\{\ddot{c}\}_{n+1} = \{\dot{c}\}_n + [(1 - \alpha)\{\ddot{c}\}_n + \alpha\{\ddot{c}\}_{n+1}]\Delta t$$

$$\{c\}_{n+1} = \{c\}_n + \{\dot{c}\}_n\Delta t + \left[\frac{1}{2} - \beta\right]\{\ddot{c}\}_n + \beta\{\ddot{c}\}_{n+1}\right](\Delta t)^2$$

Which is known as the Newmark direct integration method [29]. The parameters $\alpha$ and $\beta$ control the accuracy and stability of the solution, and are often adjusted for different time marching techniques.

Unlike many FEA models, these particular simulations were not concerned with stress or other failure criteria (such as heat transfer concerns). The main purpose of the FEA model was to determine the dynamics of the structure within an FSI context. One form of the full equation of motion in matrix form can be seen below:

$$\mathbf{M}\ddot{\mathbf{a}} + \mathbf{C}\dot{\mathbf{a}} + \mathbf{K}\mathbf{a} + \mathbf{f} = 0$$

Further simplifications can be made to this equation for cases where there is no damping in the system, or no external forces [24]. Derivations of the mass and stiffness matrices vary greatly depending on the location and type of the element, but general solutions can be found at [23,24]. One particular case in this thesis occurs when there is no structural damping, and only rigid motion is considered. The matrix equation for the e\textsuperscript{th} element is:

$$\mathbf{V} = \lambda_i \mathbf{n}_i \mathbf{n}_i^T + \lambda_{II} \mathbf{n}_{II} \mathbf{n}_{II}^T + \lambda_{III} \mathbf{n}_{III} \mathbf{n}_{III}^T$$
\[ [K^{(e)}]\{u^{(e)}\} = \{f^{(e)}\} + \{Q^{(e)}\} \]  \hspace{1cm} (15)

Which is derived from the Poisson equation [29]. The forced displacement from the assembled element matrix Q handles prescribed motions, which is done for the MAV simulations in this thesis.

FSI couples both the FEA and CFD solvers, and was the focus of the simulations for this thesis. The reasoning behind modeling FSI simulations for MAVs is that the deformation of the wing itself is too large to ignore and model accurately. As this deformation affects the surrounding fluid flow, and the fluid flow affects the amount of deformation, coupling the solvers together results in a more accurate solution. FSI can be used to couple temperature and heat flux data, but this is not necessary for the present analysis. This focuses on obtaining the fluid pressure on the surface from the fluid (SC/Tetra) solver and the displacement of the surface from the structural (ABAQUS) solver.

There are a few important details on how these software programs work together in FSI. First, it is considered a “weak” coupling between the two solvers. This simply means that each solver solves the physical quantities of their domain individually, rather than combining the physical field equations together. The residuals of the physical quantities are monitored by the solvers to ensure convergence at each time step. Next, it is considered a bi-directional coupling between the solvers. This is what allows the deformation of the structure to affect the flow field and vice versa. Unidirectional coupling is useful when the respective fields are only affected one way, i.e., when the deformation of the structure is negligible or the force from the fluid is negligible. The bi-
directional coupling permits the proper adjustments in the calculations when the
deformation is changed from the fluid flow, and subsequently the flow field is updated
from this deformation. Next, an iterative approach is used to solve the coupled equations.
This approach is illustrated in the figure below:
Figure 2.1: Diagram showing iterative FSI methodology.
While this diagram may look relatively complicated, it is quite important to the convergence of the simulations. Essentially, ABAQUS will initially calculate a displacement of the coupled regions, iterating internally until it has converged. It will then send displacement data to the SC/Tetra solver where it will deform the mesh and calculate the fluid flow for the entire domain. This data is then sent back to the ABAQUS solver with the pressure load on the coupled regions. ABAQUS then loops back and calculates adjusted displacements before going back to the SC/Tetra solver with updated displacements. The pressure flow field is calculated again with the loop continuing up to twenty times (per time step) or until convergence criteria has been met. The convergence criteria for these simulations was set at a residual (difference between value at one time step and the next) of 1E-4 for the pressure variable. While time-consuming, this helps assure the coupled simulations work together properly for an accurate solution. One important note about the data transfer between these solvers is the individual meshes do not need to be conformal. A simple inverse distance weighting method is used by each solver to interpolate data at points along the surfaces of interest [23,25]. This applies a weighted average to the inverse distance between the points of data being sent and those points at which the data is received.

The last important consideration for FSI is the under-relaxation of the mesh movement. This is handled through the SC/Tetra solver with the Gauss-Seidel under-relaxation method. This method for limiting overestimation of surface displacement uses the following equation (where \( n+1 \) is the time step and \( k+1 \) is the iteration):

\[
d^{n+1}_{k+1} = \omega \bar{d}^{n+1}_{k+1} + (1 - \omega)d^n_k^{n+1}
\]  

(16)
The parameter $\omega$ can be adjusted as needed to help reduce mesh movement. This can be helpful at times when convergence is an issue due to excessive displacements between steps. Reducing the under-relaxation parameter $\omega$ can greatly reduce errors arising from overestimated displacements from the structural solver [25]. It should be noted the domain for $\omega$ is $0 < \omega \leq 1$, where 0.1 was found to be satisfactory for these simulations. Choosing values too close to 1 could lead to a result that is not fully converged, so monitoring of the residuals was necessary.
3. VERIFICATION OF FSI

VERIFICATION OF FSI INTRODUCTION

A previous numerical benchmarking of an FSI simulation case by Turek et al. was chosen to be used as a verification of the FSI methodology prior to running the MAV case. The case was based on 3-D, laminar, incompressible channel flow around a cylinder, with an attached elastic beam [26]. The flow has an inlet parabolic profile and corresponds to a Reynolds number of 200 that has shown a periodic solution in previous results. This case investigates the y-direction amplitude of the end of the beam as a measure of merit. The basic geometry for this setup (with all dimensions in meters) is shown below in figure 3.1:

The thickness into the page of this model (not shown above) was 0.1 meters.

There were numerous goals of this verification case in preparation for running MAV cases in FSI. First and foremost, it allowed for one to compare the setup using...
ABAQUS and SC/Tetra with previous results. It was important that results obtained from this simulation agreed before moving on to modeling MAV simulations to be compared with experimental results. Next, it allowed for testing of various parameters to improve accuracy of the result. This includes varying the time step, mesh density, and under-relaxation techniques. Finally, it allowed for testing of the FSI techniques being used. As mentioned above in the background of FSI techniques, there are multiple methods that are used with varying degrees of accuracy and not all are appropriate for every type of simulation.

**ABAQUS (FEA) SETUP**

*Material and Section Properties*

The following material properties (corresponding to rubber-like polybutadiene) were used for the flexible attachment of the validation model:

<table>
<thead>
<tr>
<th>Part</th>
<th>Density (kg/m^3)</th>
<th>Young’s Modulus (Pa)</th>
<th>Poisson Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexible Attachment</td>
<td>1000</td>
<td>5.60E+06</td>
<td>0.40</td>
</tr>
</tbody>
</table>

It should be noted that one of the important characteristics of this verification case was the density ratio between the fluid (which was water) and the structure. At a value of 1:1, this is similar to the ratio of the density of the air (fluid) and wing (structure) for the MAV being studied. While this study does not have all the characteristics of a full validation study and is not treated as such, similar properties allowed for a certain level of calibration in preparation for the MAV cases.

Additionally, this verification only required looking at the movement of the flexible piece on the structure side. As such, only the flexible beam was modeled in
ABAQUS, as the remaining portion would be setup in CFD. The FEA sections of the validation were then straightforward, simply using a solid, homogeneous set with constant modulus.

**Boundary Conditions**

The verification case had two important boundary conditions. The first is known as the “encastre” boundary condition. This simply means that all six degrees of freedom (three displacements and three degrees of rotation) are set equal to zero. This was done to the first row of nodes attached to the cylinder. As the cylinder is rigid, these elements must remain in the same location without any twisting. To illustrate, one can see in figure 3.2 below where the encastre boundary condition was applied:

![Figure 3.2: Isometric view of encastre boundary condition.](image)

The other boundary condition is known as a z-symmetry condition. Considering the model as it is shown in figure 3.1 to be in the x-y plane, the flow moving from left to right creates eddies off the cylinder which make the attachment move up and down.
Again, due to the cylinder being rigid, the flexible attachment cannot twist (rotation about the x-axis) or rotate in and out of the page (about the y-axis). The z-symmetry boundary condition takes care of these situations; as it prevents rotation about both the x and y axes, along with stopping any displacement in the z direction. This condition must be applied the entire length of the attachment on both sides, as shown below in figure 3.3:

![Figure 3.3: Isometric view of z-symmetry boundary condition.](image)

It should be noted that while this condition is applied for simplicity at the same nodes which already have the encastre boundary condition, the encastre condition supersedes it at those locations. This is due to the extra restrictions of the encastre condition stopping all six degrees of freedom instead of just three.

**Meshing and Element Choices**

For the structure side, one setup was created after confirming reasonable results from post-processing. This is in contrast to the fluid side, where both a standard and refined mesh were created, which will be described later in this thesis. Linear brick hexahedral elements were chosen (known in ABAQUS as C3D8R), with further reasoning described in chapter two. The overall setup resulted in approximately 600 nodes and 240 elements, which is a relatively low amount compared to many other cases.
An important point that allowed the mesh to remain coarse but still accurate was the z-symmetry boundary condition described above. As there was no movement of the nodes or elements in the z direction, the brick elements are seeded such that they go the entire span of the flexible attachment in the z-direction. Six layers were used along the y direction along with forty layers in the x direction to complete the mesh. Pictures of the flexible attachment illustrating this mesh setup can be seen below in figures 3.4 and 3.5:

Figure 3.4: View of mesh in x-z plane showing the full span of the brick elements along the z direction.

Figure 3.5: Isometric view of mesh showing the layers of elements in x and y directions.
**Other ABAQUS Conditions**

A few more conditions must be setup for the FEA solver to run properly. A dynamic, implicit step is defined under which all the previously mentioned boundary conditions are set. Dynamic conditions are necessary for the motion of the flexible attachment, and an implicit method is used to solve the equations at the nodes during the simulation. The incremental time step was set to start at an initial tenth of a second with a minimum increment of ten nanoseconds (which was found to be smaller than necessary). This allows the software to automatically adjust as needed for a time step that is small enough for convergence. Specifying a minimum increment helped prevent the simulation from becoming too time-consuming, with slow simulations being indicative that mesh changes or other parameter adjustments might be needed.

**SC/TETRA (CFD) SETUP**

**Volume Region**

Setting up the CFD side of the simulations started with the creation of a master volume, whose dimensions were predefined based on Turek et al. simulations. Referencing figure 3.1 above, the dimensions for the volume region were 2.5 x 0.41 x 0.1 meters. The cylinder and flexible beam are cut out of the volume, as they are solids with no flow going through them, and all displacements of the beam are solved in ABAQUS. An isometric view of the resulting volume is shown below in figure 3.6:
Figure 3.6: Isometric view of CFD master volume.

Meshing

A coarse and refined mesh were created for comparison of this verification case. The mesh was created by defining octants with three regions of varying refinement, as shown below in figure 3.7:

Figure 3.7: Octant setup for mesh showing three regions of refinement.

As expected, the regions far away from the cylinder and attachment are the least refined, and the size of octants are halved near the region where the attachment will actually be moving through. Next, very close to the cylinder and flexible attachment, the mesh is refined further for very accurate pressure readings of the surfaces where data is being
transferred for the FSI simulation. Details on the sizing of octants for both the coarse and refined region are shown below in table 3.2:

<table>
<thead>
<tr>
<th>Element Sizing for Coarse Mesh</th>
<th>Element Sizing for Refined Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Size (mm)</td>
</tr>
<tr>
<td>Level 1</td>
<td>12.5</td>
</tr>
<tr>
<td>Level 2</td>
<td>6.25</td>
</tr>
<tr>
<td>Level 3</td>
<td>3.125</td>
</tr>
</tbody>
</table>

The resulting overall size from these setups was a 24,300 element, 27,300 node mesh for the coarse model, and a 34,700 element, 37,800 node mesh for the refined model. One other vital part of the meshing was the generation of prism layers near walls. Prism layers are important for accurately approximating boundary layer solutions and described in more detail under the MAV case. For verification, the top and bottom walls, along with the cylinder and flexible attachment walls included prism layers. Three prism layers were used near each of these surfaces, where approximately all three prism layers equaled the size of one octant in that region. This resulted in two sizes for the prism layers (per model), as the top and bottom walls are part of level 1 octants, and along the surfaces of the cylinder and attachment are level 3 octants. This is illustrated below in figure 3.8:
Figure 3.8: View of different size of prism layers along top wall and cylinder wall.

A look at the overall mesh generated taking into consideration both octant setup and prism layers is shown below in figure 3.9:

Figure 3.9: Overall CFD mesh including octant sizing and prism layers.

**Boundary Conditions**

A number of boundary conditions must be created in the SC/Tetra solver to properly run the simulation. The four types of boundary conditions used for the
verification simulations were a defined inlet velocity profile, two different wall conditions, and a static pressure outlet. First, a parabolic inlet velocity profile was defined for the inlet (to the left of the cylinder) ranging from zero meters per second at the top and bottom edges to a max of 3 meters per second in the center of the inlet surface. A plot of the velocity profile (where zero is the bottom of the portion of the inlet surface) is shown below in figure 3.10:

Figure 3.10: Velocity profile at inlet of master volume.

Next, for the outlet surface (right side of the model in the x-y plane) a zero static gauge pressure boundary condition was prescribed. This is done to ensure all pressure changes due to the movement of the flexible beam have died out by the time they reach the outer
boundary. If they have not, the simulation would force the zero pressure at the wall and a reflection pressure wave would occur. While it is reasonable to assume this volume is already acceptable having been used for previous simulations, this allows for an additional check that parameters are set up correctly. As backflow is assumed to be normal to the boundary, it should be zero for a converged solution with a significantly large master volume [27].

Finally, two wall conditions are used to run this verification case. The first is a stationary wall, used for the top and bottom surfaces of the master region. This sets all components of the velocity equal to zero at the wall, and the wall itself cannot move. For the cylinder and flexible beam, the wall condition prescribes the mesh velocity equal to the surface velocity. For the regions of the cylinder, this remains zero as it is fixed and will have a surrounding boundary layer. For the flexible beam, this remains variable as the attachment can move over time depending on the pressures of the surface caused by flow over the cylinder. Figure 3.11 below has all the boundary conditions labeled for clarity:

![Diagram showing boundary conditions](image)

**Figure 3.11:** Summary of boundary conditions for CFD side of verification case.
Other Conditions

A number of other conditions are set in the CFD solver to make this simulation run properly. The more common, important settings affecting this simulation are listed below in table 3.3:

<table>
<thead>
<tr>
<th>CFD Basic Settings</th>
<th>Laminar Flow</th>
<th>Transient Analysis</th>
<th>0.001</th>
<th>Every 0.1 seconds</th>
<th>Iterative</th>
<th>Gauss-Seidel method</th>
<th>0.1</th>
<th>0.1 seconds</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis Type</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Analysis Method</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Time Step</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLD Output</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Coupling Method</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Under-Relaxation for Mesh Displacement</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Under-Relaxation Coefficient</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Offset Time for FSI</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The analysis is kept within the laminar flow regime, as to remain consistent with the previous analyses presented in Turek et al. [26] Looking at flows past a cylinder one could note that it is feasible that there would be flow separation for this particular case. As the main goal for this verification was to match results to the previous simulations, this possibility was ignored. It is a transient analysis due to the time dependence of the flutter of the flexible attachment. Initially, the velocity at all nodes will be zero except at the inlet velocity boundary. The time step and FLD output (CFD file containing all CFD variable solutions to be post-processed at a particular time step) were adjusted based on trial and error. The time step must be small enough to allow convergence, but not so small that the simulation takes a needlessly long time to complete. Similarly, enough FLD files are needed to get valid results of the flexible beam over time, but not so many that hard drive space was being wasted. The details on the choice of FSI coupling
method and under-relaxation for mesh displacement method and coefficient are described above under FSI background and the validation results, respectively.

VERIFICATION RESULTS

Verification results were found for a total of ten different cases. Five cases were run using the coarse mesh, with the other five using a refined mesh. Cases were further divided between two different under-relaxation methods, and three different time steps. The breakdown of each of the case conditions is shown below in table 3.4:

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Coarse mesh</th>
<th>Refined Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Under-Relaxation Method</td>
<td>Linear</td>
<td>Gauss-Seidel</td>
</tr>
<tr>
<td>Time Step</td>
<td>0.002 0.002</td>
<td>0.001 0.001 0.0005</td>
</tr>
</tbody>
</table>

The cases were compared with previous results, primarily looking at the y-direction displacement of point A at the free end of the attachment (see figure 3.1). Figure 3.12 below compares the y-displacement of four cases:
The most important thing to note from this is both Gauss-Seidel and linear under-relaxation worked reasonably well. The key difference was whether or not the mesh was coarse or fine. It was chosen to move ahead at that point with Gauss-Seidel under-relaxation, but one more check was done at an even smaller time step at 50 microseconds. The results from this are shown below in figure 3.13:

Figure 3.13: Y-displacement data from four different cases.

Figure 3.12: Y-displacement data from four different cases.

The results from this are shown below in figure 3.13:

Figure 3.13: Results of y-displacement of point A at a 50 microsecond timestep.
As the maximum displacement changed less than five percent, confidence was gained that 100 microseconds was an acceptably small timestep. An overall comparison of the results at the 100 microsecond timestep is show below in table 3.5:

Table 3.5: Max displacements for various cases.

<table>
<thead>
<tr>
<th>Study</th>
<th>Max y-displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Result from previous study</td>
<td>0.03425</td>
</tr>
<tr>
<td>Linear Validation_Coarse</td>
<td>0.0289</td>
</tr>
<tr>
<td>Gauss-Seidel Validation_Coarse</td>
<td>0.0299</td>
</tr>
<tr>
<td>Linear Validation_Fine</td>
<td>0.0312</td>
</tr>
<tr>
<td>Gauss-Seidel Validation_Fine</td>
<td>0.0346</td>
</tr>
</tbody>
</table>

The overall result taken from this verification case was that mesh refinement was more critical than the type of under-relaxation method. The maximum displacements changed approximately 14% between the coarse and fine mesh, indicating that the mesh refinement was necessary. It should be noted there are limitations on how refined the mesh can be while still being feasible. This is due to both time related issues and convergence issues for FSI [28]. The coarser mesh showed a lower frequency and smaller magnitude compared to the fine mesh results. However, using the Gauss-Seidel under-relaxation and fine mesh the case was able to get within 1% of the previous results from Turek et al. This showed that our conditions and setup were reasonable and ready to be taken on to MAV simulations.
4. MAV CASE SETUP

MAV CASE INTRODUCTION

The MAV simulations revolved around taking a hovering scenario and investigating different geometries, flapping frequencies, amplitudes, and maximum rotation angles. Initially, a simplified wing geometry is simulated with a sinusoidal amplitude, maximum angle of rotation of 60º, and two different flapping frequencies of 5 and 15 Hz. The 15 Hz case was then completed again with a refined model based off of experimental cases to compare geometry effects. Finally, the refined model was then run for two cases to directly compare to experimental results. These cases were at a flapping frequency of 16.5 Hz and a maximum angle of rotation of 120º. They were run with the previous sinusoidal amplitude and also an amplitude based on a four-bar motion calculated in Matlab. This allowed for comparison of different amplitudes and also a qualitative comparison with high-speed camera footage of the experimental data.

WING GEOMETRY

*Uniform Modulus Wing*

The first wing model was a simple, 1.5 mm thick rectangular wing with a uniform modulus. This provided a first order approximation to the experimental wing. The approximate dimensions of the wing are shown in figure 4.1 below, given in millimeters. At the upper left in the figure is the wing extension and wing holder. The wing extension and holder were the same material as the wing.
Refined Model

The wing chosen to be analyzed was based off of a dragonfly and developed through local wind tunnel experiments by Dawson et al. [14]. The approximate dimensions of the wing are shown in figure 4.2 below, given in millimeters. The wing is made of a strong carbon fiber rod with relatively weaker branches, covered on both sides with Mylar film.

For use in this case, a few simplifications were made for the simulations:

- The carbon fiber branches and rod are assumed to be of rectangular cross-section and of uniform thickness. In reality the branches taper at the edges, and this was
approximated to allow for the mylar film to lay flat in this model. This helps with surface-to-surface constraints described later in this chapter.

- An extra piece of film (attached by edges to the other two pieces of film) was added along the outer edge to “seal” the wing. This is done to allow for proper control volume placement for the CFD portion of the model, described later in this chapter.

- The holder in which the strong carbon fiber is placed is assumed to be rigid. While this is not completely the case, this is done as the deformation is insignificant and does not affect the results obtained from the motion of the wing.

**ABAQUS (FEA) SETUP**

*Material Properties*

The following material properties were used for the various portions of the refined model:

<table>
<thead>
<tr>
<th>Model Part</th>
<th>Density (g/m³)</th>
<th>Young’s Modulus (Pa)</th>
<th>Poisson Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid Wing</td>
<td>1740</td>
<td>1.80E+11</td>
<td>0.25</td>
</tr>
<tr>
<td>Carbon Fiber Rod</td>
<td>1600</td>
<td>1.80E+11</td>
<td>0.3</td>
</tr>
<tr>
<td>Carbon Fiber Branches</td>
<td>1600</td>
<td>4.0E+10</td>
<td>0.3</td>
</tr>
<tr>
<td>Mylar Film</td>
<td>1200</td>
<td>3.00E+09</td>
<td>0.38</td>
</tr>
</tbody>
</table>

It should be noted that the stiffer rod attaching the body of the MAV to the wing was used to prevent excessive bending near the wing attachment. The branches of the wing are significantly more flexible to allow deformation to generate lift. While the film also
slightly stiffens up the wing, its primary purpose is also to generate lift. The pressure loads are captured by the membrane and then transmitted to the supporting structure.

Section Setup

As the rectangular wing had a constant modulus, no sections were necessary for that wing. To model the higher strength of the rod of the experimental wing, the wing is divided into different sections for the rod and branches. This is all done in ABAQUS as the CFD solver is only concerned with calculating pressures for FSI and does not look at material properties of the wing. To illustrate this, figure 4.3 below shows the two different sections of the branches:

![Figure 4.3 The two wing sections corresponding to the material properties in table 4.1.](image)

Constraints

The rectangular wing only requires two inputs to specify the motion of the model. There were four constraints used when setting up the refined model to keep the motion as realistic as possible. First, both models required the nodes around the holder of the wing to be kept rigid. This is done while defining an arbitrary reference point on the holder. This then allows the reference point to control all boundary conditions (which will be listed below) and the amplitude of the wing.
Next, both models required the connecting surfaces between the holder and the wing to be tied together. It had been shown in previous tests that without this tie constraint, the wing would simply fall out of the holder during the simulation. In actual lab tests, the wing was attached to the holder so this could not happen. The wing was specifically partitioned in ABAQUS so that only the touching surfaces would be held together as shown below in figure 4.5:

Finally, the last two constraints (for the refined model only) were to hold the branches of the wing to the film where the two surfaces are in contact. This was done to prevent the film from separating from the branches of the wing. Below in figure 4.6, one
can see the constrained surfaces where the film and branches touch on the top film. The other constraint is essentially the same but on the other side of the wing on the bottom film.

![Figure 4.6: Region where the carbon fiber branches are tied to the film.](image)

**Boundary Conditions and Amplitude**

For the structure solver, only one boundary condition needs to be applied. This boundary condition is applied to the wing holder and is used to prescribe the motion. The boundary condition is simply to limit the reference point to only rotating in the y-direction. The reference point is put along the rotation of axis so that all displacements of the point remain zero. This allows the entire rigid body to remain rotating in the same direction, resulting in all lift being in the z-direction as shown below in table 4.2:

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>x-displacement</td>
<td>0</td>
</tr>
<tr>
<td>y-displacement</td>
<td>0</td>
</tr>
<tr>
<td>z-displacement</td>
<td>0</td>
</tr>
<tr>
<td>x axis rotation</td>
<td>0</td>
</tr>
<tr>
<td>y axis rotation</td>
<td>1</td>
</tr>
<tr>
<td>z axis rotation</td>
<td>0</td>
</tr>
</tbody>
</table>
It should be noted that as this boundary condition is only applied on the rigid wing holder, it does not prevent the wing from twisting, i.e., rotating about the x-axis. The rotation around an axis is prescribed by multiplying the table value by an amplitude as follows.

Two cases were run to compare the amplitude effects, which were a sinusoidal motion and a theoretical four-bar motion derived from Matlab calculations. The first simulation represented the wing going through a 60° rotation, and started from the “top position,” which is rotated 30° in the positive y direction. The other simulation was rotated 60° for the Matlab calculations simulations. The 60° was chosen to compare to experimental testing (the four-bar linkage can change sizes to affect the overall amplitude). Each model was rotated to the initial position where the velocity was zero before the simulation began.

The Matlab data was created using the equations of a four-bar linkage outlined in Smith [30]. The sizes for the links were based entirely off their experimental data, and 16.5 Hz was used for direct comparison to experimental results. The plot of two cycles of the amplitude based on their four-bar Matlab simulation can be seen below in figure 4.7:
Figure 4.7: Four-bar simulation data calculated in Matlab.

This data was very similar to an actual sinusoid, and was expected to give the best approximation to experimental results.

Meshing and Element Choices

The solid structure was meshed using two element types. A solid, 20-node quadratic brick (3-D stress element) was used for the holder and wing branches. Reduced integration, which is commonly used for these elements to reduce computational time, was turned off to improve accuracy of the result. The quadratic elements are useful as they rarely exhibit shear locking, a common problem with first-order, linear elements. Shear locking typically occurs in pure bending situations (which could occur as the wing is fixed at one end and free to flex on the other) where the element records incorrect nonzero shear stress [23]. To use the automatic meshing tool within the ABAQUS software, multiple partitions were created to prevent distorted elements from forming.
The other element type used was the shell element, necessary for the thin layer of film on the wing. The particular shell elements used were linear quadrilateral elements along with linear triangular elements. Among other shell element choices, these tend to have a more accurate solution and are less prone to membrane and bending hourglass issues [23]. These shell elements use finite membrane strains, with a 10 μm thickness specified based on experimental wings [14]. Due to the irregular shape of the film, it is necessary that the triangular elements be specified for regions where the quadrilateral elements would be too distorted for accurate results.

A look at the mesh of the wing model is shown below in figures 4.8 and 4.9:

Figure 4.8: Mesh of the holder with brick elements.
Figure 4.9: Mesh of the experimental wing with both brick and shell elements.

More information on the element choices is contained in the background chapter of this thesis.

*Other ABAQUS Conditions*

A few more conditions must be setup for the FEA solver to run properly. A dynamic, implicit step is defined under which all the previously mentioned boundary conditions and amplitude are set. Dynamic conditions are necessary for the motion of the wing, and an implicit method is used to solve the equations at the nodes during the simulation. The incremental time step is set to start at ten microseconds with a minimum increment of ten nanoseconds. Just like the validation case, this allows the software to automatically adjust as needed for a time step that is small enough for convergence. Specifying a minimum increment helped prevent the simulation from becoming too time-consuming, with slow simulations being indicative that mesh changes or other parameter adjustments might be needed.

**SC/TETRA (CFD) SETUP**
Volume Regions

Setting up the CFD side of the simulations started with the creation of a master volume, which is much larger than the dimensions of the wing. The region must be at least 3-5 times the width of the wing in every direction to avoid boundary condition errors. This could happen if atmospheric pressure was defined at the outer walls of the simulation, but it was not far enough away from the wing to where there would be pressure changes due to the motion of the wing. This is described in more detail below under boundary conditions.

For the original sinusoidal simulations, a rectangular prism was used as the volume region. When results appeared to have some errors due to boundary condition effects, the model was updated to a hemispherical model shown below in figure 4.10:

Figure 4.10: Hemispherical volume used for master region in simulations.

The hemisphere had a diameter of 2 meters, making the radius of 1 meter sufficiently large for this size of wing. Use of this region made for a mesh of approximately 1.12 million elements and approximately two hundred thousand nodes. The mesh is very
refined near the wing, getting coarser as it approaches the outer boundaries to minimize the number of nodes.

One addition for the realistic amplitude case was the inclusion of the tail of the MAV to the simulation. The tail of the MAV is important in helping control the maneuverability of the MAV during flight. As these simulations are only looking at a hovering case and focused on wing characteristics, the tail is just a part of the fixed master volume. While this does not affect the simulation in any way computationally, it allows the viewer to get a better idea of how the wing is oriented with the rest of the MAV. Other than the shape of the tail seen easily in the y-z plane, there is a horizontal stabilizer which extends in the x-direction. The tail within the master region in the y-z plane is shown below in figure 4.11 (wing not shown):

![Diagram of y-z plane view of master region with tail included.](image)

Figure 4.11: y-z plane view of master region with tail included.

Next, a smaller rectangular prism region was created which contains the wing. Again, details of why two regions were chosen around the wing are described below in
overset mesh conditions. An isometric view of the rotating region can be seen below in figure 4.12:

![Isometric view of rotating region.](image)

Figure 4.12: Isometric view of rotating region.

An important note on meshing the wing is that a thin prism layer is used to improve accuracy very close to the wall conditions of the wing. Two layers of prisms were used, with a thickness of 0.5 millimeters. Prism layers help deal with boundary layers formed near the edge of the wall. Their smaller elements allow for calculating the magnitude of the wall friction, helping determine overall resistance to the flow path [25]. No prism layers are inserted within the wing, as all displacements of the wing itself are calculated using the FEA solver (and obviously there is no fluid within the wing to create a boundary layer there). A cross sectional view of this part of the mesh with the two prism layers on each of the wing can be seen below in figure 4.13:
Figure 4.13: Cross-sectional view of rotate region, noting 2 prism layers near wall of wing.

One can see that the mesh is very refined near the wing, but gets a bit coarser towards the edge of the rectangular prism. The rectangular prism was made large enough to allow for 2-3 layers of the larger elements to maintain smooth results. Details of the sizing of octants for element meshing are listed below in table 4.3:

<table>
<thead>
<tr>
<th>Element Sizing for mesh</th>
<th>Type</th>
<th>Size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Prism Layers</td>
<td>0.5</td>
</tr>
<tr>
<td></td>
<td>Refined rotating region</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>Coarse rotating region</td>
<td>8</td>
</tr>
<tr>
<td></td>
<td>Refined master region</td>
<td>8</td>
</tr>
<tr>
<td></td>
<td>Coarse master region</td>
<td>64</td>
</tr>
</tbody>
</table>

Making the rotating region any smaller (i.e., closer to the wing) does not allow for enough larger elements to be present for a smooth transition into the master region. This results in discontinuities in the mesh, and undesirable inaccuracies with the solution.

Boundary Conditions
A number of boundary conditions must be created in the SC/Tetra solver to properly run the simulation. The three types of boundary conditions used for these simulations are free slip walls, defined pressure, and a wall condition. First, the free slip wall is a simple “do nothing” boundary condition applied to the tail and symmetry wall of the simulation. Any pressure, velocity, or other parameter reaching a free slip wall will simply continue out of the domain and have no effect on the remainder of the simulation. The tail, as mentioned above, is added purely for cosmetic reasons and so particles are able to flow straight through it. The symmetry wall, which is the flat portion of hemisphere, is used in post-simulation to replicate modeling both wings at the same time. It should be noted that in real life the wings would not have perfect symmetry and the tail would affect flight characteristics. Any pressures or velocities calculated in this region are not necessary for determining lift, thus the use of a free slip wall.

Next, for all curved sides of the master volume a pressure of one atmosphere is prescribed. This helps determine if the master region was significantly large to capture all effects from the motion of the wing. If for some reason the region was too small, the simulation would force the static pressure to be zero at the curved surfaces. This would then result in a reflection pressure wave and change the results of the simulation.

Finally, a wall condition is prescribed for all outer surfaces of the wing. More specifically, the wall condition is where the mesh velocity is equal to the wall velocity. This sets the initial mesh movement, as the simulation starts with the motion prescribed from the FEA solver. For clarity, all boundary conditions are labeled below in figure 4.14:
Overset Mesh

One of the most important parts of this type of simulation for CFD is using an overset mesh. A standard deformable mesh is not suitable for accurate results due to the large range of movement of the wing. An overset mesh, on the other hand, uses the two volume regions described above to handle both the range of motion and deforming geometry. While meshed separately, the rotating region is placed inside the master region to make two overlapping, unstructured grids. The key part of making these meshes work together is that the outer edges of the rotating region have element sizes approximately equal to the element sizes of the master region within the area that it is rotating [25]. This is the reason that the master region has very fine parts of the mesh where the wing itself passes through, and gets coarser farther away from the wing. While it would be feasible that the entire master volume region could all be of equal refinement, significant simulation time is saved by reducing the refined region to only where it is absolutely necessary. To illustrate this, one can look below and see in figure 4.15 how the mesh changes farther away from the wing and rotating region:
Other Conditions

A number of other conditions are set in the CFD solver to make this simulation run properly. The more common, important settings affecting this simulation are listed below in table 4.4:

Table 4.4: Basic CFD settings

<table>
<thead>
<tr>
<th>CFD Basic Settings</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis Type</td>
<td>Turbulent Flow</td>
</tr>
<tr>
<td>Turbulent Flow</td>
<td>RANS</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Standard k-EPS model</td>
</tr>
<tr>
<td>Analysis Method</td>
<td>Transient Analysis</td>
</tr>
<tr>
<td>Time Step</td>
<td>0.001</td>
</tr>
<tr>
<td>FLD Output</td>
<td>Every 2 Cycles</td>
</tr>
<tr>
<td>Coupling Method</td>
<td>Iterative</td>
</tr>
<tr>
<td>Under-Relaxation for Mesh Displacement</td>
<td>Gauss-Seidel method</td>
</tr>
<tr>
<td>Under-Relaxation Coefficient</td>
<td>0.1</td>
</tr>
</tbody>
</table>

One other important property is that of the fluid through which the wing passes. The important difference between moving the wing in FEA and adding CFD for an FSI simulation is that the fluid can now be defined (whereas ABAQUS will default to a
vacuum simulation otherwise). For this case it is just air at a steady temperature of 20º Celsius with fluid properties defined below in table 4.5:

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Density (kg/m^3)</th>
<th>Viscosity (Pa-s)</th>
<th>Cp (J/(kg-K))</th>
<th>Thermal Conductivity (W/(m-K))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air</td>
<td>1.206</td>
<td>1.83E-5</td>
<td>1007</td>
<td>0.0256</td>
</tr>
</tbody>
</table>

The highly separated flow (due to the wing changing directions) warrants the use of a turbulence model for these simulations. Details on the specific turbulent flow and turbulence model are described under the CFD background information earlier in this thesis. It should be noted a Reynolds number is not calculated due to the changing length of the wing in both span-wise and chord-wise directions. Additionally, the nature of a hovering case this simulation does not have an inlet velocity from any direction (which would make it possible to choose a characteristic length based on the direction of the flow.) The analysis is considered to be transient due to its time dependence, i.e., the wing moves with time and how it gets to its position is important to the overall solution. This is in contrast to a steady-state solution, where the method to get to the final solution is irrelevant to the overall solution [25].

The time step and FLD output were adjusted through trial and error. Choosing a time step too large will result in divergence as the case will not be able to find a particular solution if too much time passes between each calculation. On the other hand, choosing a time step too small will result in the simulation taking an unnecessarily long time to solve. The FLD files were adjusted based on desired results balanced with available hard drive space. Keeping FLD files every time step was deemed unnecessary considering the small time step already defined. However, keeping less data (for one simulation every 5
cycles or 0.005 seconds was chosen) resulted in choppy video results and too much missing data between each cycle for an accurate solution. The details on the choice of FSI coupling method and under-relaxation for mesh displacement method and coefficient were described previously under FSI background and the validation results, respectively.
5. RESULTS AND DISCUSSION

RECTANGULAR WING SIMULATIONS

The first cases all involved the rectangular wing model, primarily for the simpler wing allowing for comparison to more realistic cases. Two cases were run in FSI, at 5 Hz and 15 Hz flapping frequency. The post-processing involves looking at all pressure forces on the wing in the z-direction, and is used to determine lift numbers. Attached below, one can see the lift history for approximately 2.5 cycles of the 5 Hz rectangular wing model:

![Lift Force Graph](image)

Figure 5.1: Lift history of 5 Hz rectangular wing model.

A few things can be noted from this case. First, one can see that the lift history is not particularly smooth, and might even look like the wing is vibrating in a strange
manner. The time and space residuals were first checked to ensure the simulation was converging at each time step before continuing. A few snapshots of the wing in motion were taken to investigate further:

Figure 5.2: Pressure contours at starting position (0.2 seconds).

Figure 5.3: Pressure contours at 0.24 seconds.

Figure 5.4: Pressure contours at 0.28 seconds.
It should be noted that all of these snapshots looked at pressure contours of the wing between 1 and -1 Pascals. The snapshots themselves do not reveal any vibration, however, there is very little deformation at the peaks of the cycles. A modal analysis based on these simulations (using just the structural solver) did not yield any realistic results. However, considering the similarities of the wing to the analysis in [33], it seems feasible that the wing is responding to a natural frequency in the 50-60 Hz range. The very low pressure differences are shown visually by the lack of deformation of the wing.
The next comparison investigated for this wing was to increase the frequency to 15 Hz. First, this would allow a sanity check to ensure that the pressure forces at 15 Hz should be considerably higher than those at 5 Hz. Additionally, the higher frequency is much closer to what real wing experiments ran at in testing. Shown below in figure 5.7 one can see the lift history of the rectangular wing at 15 Hz:

![Lift history for 15 Hz of rectangular wing.](image)

One of the more unusual results is the inconsistent peaks on various cycles. This could be an indication of a multimodal structural response where the total deformation is the sum of a few low order modes. This is sometimes seen when a structure is excited by a broadband source such as a random acoustic load. There isn’t an applied acoustic load in this case, but there are non-uniform distributed loads due to the pressure. The constant modulus wing could be a reason for these results. In nature, the tips of the wing deform
significantly more than portions of the wing closer to the base. However, for a constant modulus the wing tends to deform consistently span-wise. In reality, the wing cannot all deform the same amount due to the constraints of the wing to the base. Higher velocities amplifies the effect of the constant modulus, causing the variation in pressure peaks shown above. A look at both lift histories together is shown below in figure 5.8:

![Lift Force Comparison](image)

Figure 5.8: Lift history for half a second of 5 and 15 Hz of the rectangular wing.

EXPERIMENTAL WING MODEL SIMULATIONS

Next, it was time to look at the FSI results using the model of the experimental wing. To compare with realistic cases, the simulation was run at 15 Hz. The first iteration used the sinusoidal amplitude defined in ABAQUS. Results of the lift
generation (again found using pressure forces on the surface of the wing in the z-direction) are shown below in figure 5.9:

![Lift Force Graph]

Figure 5.9: Lift history of experimental wing model at 15 Hz.

A key difference in these results are the consistent peaks shown throughout each cycle. While there are some unusual spikes at some of the cycles, this could be due to a variety of factors. One common issue found when modeling complex geometry was the resulting mesh. Even with sufficient refinement, there are often small areas that do not mesh well and have a lower orthogonal quality. This, in turn, results in numerical error in the overall simulation, which is what is most likely seen in the inconsistent spikes above. It remains that the overall result is much smoother than that of the rectangular wing simulation. This was expected with a variable modulus wing that can more easily have greater deformations near the tip of the wing, as observed in nature. Snapshots of
the wing during various portions below in figures 5.10-13 further confirm the smooth transitions during the cycle:

Figure 5.10: Pressure contours on wing mid down-stroke at 0.016 seconds.

Figure 5.11: Pressure contours on wing at bottom of cycle at 0.032 seconds.
Figure 5.12: Pressure contours on wing mid up-stroke at 0.048 seconds.

Figure 5.13: Pressure contours on wing near the start of its cycle at 0.074 seconds.

These snapshots are of a cross section going straight through the wing, giving a clear look at the 2-D pressure generated from the movement of the wing. This showed continuity between the two meshes used for overset by confirming no unusual pressure changes between the inner and outer mesh boundaries. Additionally, pressure seemed physically realistic, with positive pressure in front of the wing motion and negative pressure behind it.

Two more cases were completed for comparisons to experimental wing results. These cases used a larger amplitude of 120º rather than the 60º of rotation of previous
cases. One case was completed using the sinusoidal amplitude used in the previous cases, while the final case used the Matlab-based four-bar simulation. The lift force history of the sinusoidal case is shown below in figure 5.14:

![MAV 16.5 Hz Sine case](image)

**Figure 5.14:** Lift history of sinusoidal, 120° amplitude case.

It should be noted that the lift is much greater for these cases than previous ones due to the changes in flapping frequency and greater amplitude range. This case showed very reasonable results, with a repeatable motion. While it is uncertain whether the peaks at each minimum and maximum point are realistic, they do not appear to be a varying numerical error as it occurs at each transition from upstroke to downstroke and vice versa. Finally, the case simulating the four-bar motion generated in Matlab was run, with the lift generation history shown below in figure 5.15:
This case showed very similar results to that of the previous sinusoidal case, as expected. The cases showed very similar maximum magnitudes, with 0.1614 and 0.1644 for sinusoidal and Matlab amplitude, respectively. The differences in magnitude that can also be seen through comparing the peaks of the two above figures can be attributed to the nature of the four-bar motion. A look at the two cases superimposed to each other can be seen below in figure 5.16:
Figure 5.16: Side by side comparison of four bar and sinusoidal motion.

The last comparison made was between the flight path of the lab data and that of the simulated 16.5 Hz, Matlab-defined four-bar motion case. The experimental data taken by Smith recorded high-speed camera footage at various time intervals [30]. A look at a few of those points along with the modeled wing data are shown below in figures 5.17-20:
Figure 5.17: High speed camera footage and simulation at beginning of cycle.

Figure 5.19: High speed camera footage and simulation at mid up-stroke.

Figure 5.18: High speed camera footage and simulation at upper end of stroke.
The comparison to experimental data shows the biggest drawback from these simulations – the wing does not twist along the axis nearly as much as in experimental runs. This could be for a number of reasons, but is most likely due to the limited accuracy of the stiffness used along the length of the wing. It appears to be much weaker experimentally to allow for significantly more deformation at the ends of the wing.

To gather more information on this matter, a few extra cases were considered. First, the 16.5 Hz, matlab prescribed motion was run again with a Young’s modulus reduced by a factor of 10 in the branches. Unfortunately, the FSI solver had difficulty converging for higher levels of deformation. To combat this, the Young’s modulus was brought back up to a factor of 5 less than that listed in table 4.1. Again, only the modulus of the branches was changed as this will have the greatest effect of deformation away from the wing holder. While this case was also unsuccessful, two more cases with reduced time steps were attempted. Even with an order of magnitude smaller time step (100 microseconds), the simulation was not able to converge for one full cycle. However, a look at the wing’s position shortly before divergence does show significant deformation:

Figure 5.20: High speed camera footage and simulation at mid down-stroke.
As one can see, there is both span-wise bending and twisting of the wing, even on just the first downstroke. With a converging solution, it seems likely the simulation would show even more twisting, and follow experimental data quite well. As the divergence error was due to the CFD solver registering a negative volume error, it could be due to a number of reasons. It is possible the time step would need to be even lower or some meshing changes could help improve solver convergence. It is recommended some type of a cluster or supercomputer be used to investigate smaller time steps, as one second of data with eight cores would have approximately one month of computation time at the 1E-4 second time step. Further studies into improving this result are beyond the scope of this thesis, but it is something the author hopes will be investigated by future students and researchers.
6. CONCLUSION

The up and coming field of MAVs has plenty of room for growth in the future. There is a plethora of parameters such as size, power, weight, and maneuverability. They also have a number of different functions in different environments including surveillance, military reconnaissance, and emergency response. To meet the desired goals in these areas, a number of challenges must be overcome for MAVs to be a successful research endeavor.

The purpose of this thesis was to demonstrate the modeling of a wing of a MAV using Fluid-Structure Interaction techniques. To begin this process, a case verifying the FSI solvers was chosen to be representative of a similar but simplified model. The case chosen from Turek et al. looked at flow over a cylinder with a flexible attachment. The y-displacement of the model with the refined mesh and Gauss-Seidel under-relaxation showed agreement with previous results within approximately one percent.

Upon completion of the verification, a number of varying cases were considered using FSI. Two different wing models were created looking at a constant versus a variable modulus wing. In addition, two different amplitudes were used to model the wing in a hovering scenario. Each of the amplitudes were applied to the rigid wing holder, which resulted in the motion transmitting through to the flexible wing. The first amplitude was a general sinusoidal motion, while the second was created based on a theoretical four-bar motion calculated via Matlab.

The results from the simulations first confirmed the feasibility of using FSI to model MAV wings in flight. It is important to note that with the help of modern
computer processing power, the ability to do this research has only occurred in the last few years. Even with parallel processing of up to eight cores, the simulations averaged around two weeks to run. The results demonstrated the importance of the variable modulus wing over a constant modulus wing. While a variable stiffness wing was expected to show better results (and is, after all, the type of wing insects have in nature), it served as a sanity check to continue with more complicated simulations. The investigation of varying frequencies helped show another interesting point: one can determine a minimum frequency at which the wing will no longer flap smoothly (and generate lift effectively). This is affected by both the range of motion and the frequency of the wing. The different amplitudes allowed for a comparison of a simple sinusoidal wing to that of a four-bar motion to determine if there was significant difference. As data showed results maximum magnitude within 2% while generating a very similar lift curve versus time, it yields that the sinusoidal model is a good estimation of the four-bar motion for these cases. The final comparison to lab results showed how the model followed experimental data. While the experimental data showed significantly more deformation, initial simulations of a more compliant wing demonstrated both twisting and span-wise bending seen in experiments. Although the cases did not converge completely, it showed feasibility for continued research with these techniques. Future studies could look into adjusting material properties to achieve a more accurately modeled flight path. Additional research could be done looking at different element types to yield a correct modal analysis. With verified modal natural frequencies, one could consider wing design issues from a structural dynamics perspective as well with this FSI setup.
FSI techniques have been proven to be an important addition in to many CFD simulations, and MAVs are no exception. Aided by ever increasing computer power, the time it takes for many of these simulations could be greatly reduced. It is hoped that a reader looking to continue this research has a number of advancements through which he or she can continue. For instance, cases could be examined for a variety of different wing flights; such as turning, elevation changes, and coping with various surrounding air conditions. Research could also continue through exploring different amplitudes, or modeling different wing setups. As FSI techniques become more commonly used, it should be easier to create parametric studies with adjustments to wing geometries (or any of the other aforementioned settings of a model) for running batches of simulations. This should allow CFD analyst engineers to help save both time and money for experimental researchers, by quantifying lift generation of wings more accurately. This allows for more optimization of MAVs as a whole, and could potentially open them up to even more not yet imagined opportunities.
REFERENCES


   http://www.bakker.org/dartmouth06/engs150/06-bound.pdf


