AERODYNAMIC EXPERIMENTS OF A DUAL DUCTED FAN VEHICLE
IN HOVER AND EDGEWISE FLIGHT

A Thesis in
Aerospace Engineering

by
Ryan Franklin Hook

©2011 Ryan Franklin Hook

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Master of Science

August 2011
The thesis of Ryan Franklin Hook was reviewed and approved* by the following:

Dennis K. McLaughlin  
Professor of Aerospace Engineering  
Thesis Advisor

Sven Schmitz  
Assistant Professor of Aerospace Engineering

James J. Dreyer  
Senior Research Engineer and Head  
CFD Analysis Dept.  
Applied Research Laboratory  
The Pennsylvania State University

George A. Lesieutre  
Professor of Aerospace Engineering  
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School
ABSTRACT

Aircraft capable of vertical takeoff and landing, as well as hover, have a wide variety of defense and commercial applications. Unrestricted by runway access and the ability to hover on location for extended periods of time allow the vehicle to complete missions impossible for fixed-wing aircraft. In general, helicopters have filled this role, but new aircraft are continuously analyzed in search of an improved alternative aircraft.

Recently, a class of unmanned air vehicles (UAVs) has utilized a single-ducted fan to provide lift and propulsion. Compared to the isolated rotor of a helicopter, placing a duct around the rotor increases the total thrust of a vehicle, allowing for the vehicle size to decrease while increasing payload capacity. The vehicle safety is also improved when operating in close quarters by shielding the rotors from strikes. There is also potential for reduced noise with the proper acoustic shielding applied to the duct. However, undesirable aerodynamic characteristics are also associated with the addition of the duct when the vehicle enters forward flight.

High drag and large nose-up pitching moments are the two leading aerodynamic flaws experienced by a ducted lift fan as it enters forward flight and encounters edge-wise flow. A ducted fan wind tunnel model was designed and fabricated for the purpose of quantifying these unfavorable aerodynamic characteristics. Instead of a single-ducted fan vehicle, this research explores the unique concept of a tandem dual-ducted lift fan vehicle, which would greatly expand the payload capabilities over a single-fan.

Measurements were first obtained from the dual-ducted fan model while operating in hover. Rough profiles of the velocity magnitudes above and below the forward and aft fans were obtained through measurements made with a mini-vane anemometer. The velocities were shown to be more ideal in the aft duct. The flow in the forward duct was further examined with a five-hole pitot probe above and below the rotor, allowing for measurement of the velocity magnitude and the three velocity components. The axial velocity was found to be the dominating velocity
component above and below the rotor, while the down-stream and cross-stream components related to the slipstream contraction above and below the rotor. The total thrust of the model was measured with the use of a force balance and the max thrust coefficient was found to be 0.022.

With the use of Penn State’s wind tunnel facilities, forward flight was simulated. Flow visualization performed with a single, vertical smoke wire revealed flow separation over the leading edge radius. Velocity magnitude measurements from a five-hole probe and kiel probe also recorded flow separation in the front half of the forward duct. The five-hole pitot probe also measured highly angled flow through the duct. A force and moment balance measured the aerodynamic lift, drag, and pitching moment in several forward flight configurations.

The need for a computation tool for the design and analysis of these vehicles was recognized. Working with Dr. James Dreyer of the Applied Research Lab at Penn State, two types of simulations were designed and validated. The first model was a steady state simulation which used a momentum source in place of each rotor to simulate rotor thrust. While this proved capable of rapid-turnaround times, it was unreliable at higher forward flight velocities. The second model was an unsteady, time-dependent simulation which resolved the rotating rotors. This fully-resolved model returned lift and drag forces within 5% of the experimental values but demanded greater time resources in order to reach a steady state.

A streamlined nose was fitted over the blunt nose of the original dual-ducted fan wind tunnel model in an attempt to reduce overall drag and flow separation off the leading edge radius. Flow around the streamlined model was analyzed with the use of smoke wire flow visualization and five-hole probe and kiel probe measurements within the forward duct. Decreased flow separation in the forward duct was made evident by the probe measurements due to increased flow velocities near the leading edge radius. Compared to the baseline blunt model, the drag was decreased by 4% in some cases and nose-up pitching moment was increased for most conditions.
TABLE OF CONTENTS

LIST OF FIGURES ........................................................................................................................................ VII
LIST OF TABLES ........................................................................................................................................... XII
NOMENCLATURE ........................................................................................................................................ XIII
ACKNOWLEDGEMENTS ........................................................................................................................... XV

CHAPTER 1 INTRODUCTION ........................................................................................................................... 1
  1.1 BACKGROUND AND MOTIVATION ......................................................................................................... 1
  1.2 DUCTED FAN ADVANTAGES .................................................................................................................. 5
  1.3 DISADVANTAGES .................................................................................................................................. 7
  1.4 BLADE AND DUCT DESIGN ................................................................................................................ 8
  1.5 OBJECTIVES ....................................................................................................................................... 11
  1.6 THESIS OVERVIEW .......................................................................................................................... 12

CHAPTER 2 DESCRIPTION OF WIND TUNNEL MODELS .............................................................................. 14
  2.1 DUAL FAN MODEL .............................................................................................................................. 14
      2.1.1 Fuselage / Duct Design ................................................................................................................ 14
      2.1.2 Rotors ........................................................................................................................................ 18
      2.1.3 Motor and Electronics .............................................................................................................. 20
  2.2 STREAMLINED NOSE ......................................................................................................................... 23
      2.2.1 Design and Fabrication ............................................................................................................. 23
      2.2.2 Attachment ................................................................................................................................ 25

CHAPTER 3 EXPERIMENTAL TECHNIQUES AND FACILITIES ....................................................................... 27
  3.1 MINI-VANE ANEMOMETER ............................................................................................................... 27
  3.2 FIVE-HOLE PITOT PROBE ................................................................................................................... 28
      3.2.1 Description .................................................................................................................................... 28
      3.2.2 Calibration ..................................................................................................................................... 29
      3.2.3 Post-Processing .......................................................................................................................... 36
  3.3 KIEL PROBE ....................................................................................................................................... 37
      3.3.1 Description .................................................................................................................................... 37
      3.3.2 Calibration ..................................................................................................................................... 38
      3.3.3 Post-Processing .......................................................................................................................... 41
  3.4 HAMMOND FACILITIES ....................................................................................................................... 42
      3.4.1 Hammond Wind Tunnel .............................................................................................................. 42
      3.4.2 Hammond Force and Moment Balance ..................................................................................... 43
  3.5 ACADEMIC PROJECTS BUILDING FACILITIES .................................................................................. 46
      3.5.1 Academic Projects Building Wind Tunnel ................................................................................ 46
      3.5.2 Academic Projects Building Force and Moment Balance ....................................................... 47

CHAPTER 4 BASELINE MODEL RESULTS AND DISCUSSION ....................................................................... 51
  4.1 HOVER ............................................................................................................................................... 51
      4.1.1 Flow Measurements ..................................................................................................................... 51
      4.1.2 Force and Moment Measurements .............................................................................................. 61
  4.2 FORWARD FLIGHT ............................................................................................................................. 64
      4.2.1 Flow Visualization ...................................................................................................................... 64
      4.2.2 Flow Measurements .................................................................................................................. 67
      4.2.3 Force and Moment Measurements .............................................................................................. 74
CHAPTER 5 BASELINE DUAL DUCTED FAN COMPUTATIONAL FLUID DYNAMICS .......................................................... 84
5.1 MOMENTUM SOURCE .......................................................................................................................... 86
5.2 FULLY RESOLVED ROTATING MESH .................................................................................................. 90
CHAPTER 6 COMPARISON OF THE STREAMLINED MODEL TO THE BASELINE ................................................. 96
6.1 FLOW VISUALIZATION ..................................................................................................................... 97
6.2 FLOW MEASUREMENTS ................................................................................................................... 98
6.3 FORCE AND MOMENT MEASUREMENTS ....................................................................................... 102
CHAPTER 7 CONCLUSIONS AND SUGGESTIONS FOR FUTURE WORK ................................................................. 113
7.1 SUMMARIZING CONCLUSIONS ......................................................................................................... 113
7.1.1 Hover and Forward Flight Experiments of a Baseline Dual-Ducted Fan Model ...................... 113
7.1.2 Design and Validation of CFD Codes for a Dual-Ducted Fan ................................................. 117
7.1.3 Forward Flight Experiments of a Streamlined Dual-Ducted Fan Model ............................ 118
7.2 SUGGESTIONS FOR FUTURE WORK ............................................................................................... 121
7.2.1 Baseline Model ....................................................................................................................... 121
7.2.2 Computational Fluid Dynamics .............................................................................................. 125
REFERENCES ........................................................................................................................................... 127

APPENDIX A MATLAB CODE AND RESULTING CALIBRATION FILE FOR FIVE-HOLE PITOT PROBE CALIBRATION ........................................................................................................ 130
A.1 RAW CALIBRATION FILE FORMAT ............................................................................................... 130
A.1.1 File Format .............................................................................................................................. 130
A.1.2 Example File .......................................................................................................................... 131
A.2 MATLAB CODE ........................................................................................................................... 131
A.3 RESULTING CALIBRATION FILE ................................................................................................. 133

APPENDIX B MATLAB CODE AND FILE FORMATS FOR PROCESSING FIVE-HOLE PITOT PROBE DATA FROM AN UNKNOWN FLOW .............................................................................................................. 135
B.1 FILE FORMAT FOR RAW DATA OF UNKNOWN FLOW ........................................................................ 135
B.1.1 File Format .............................................................................................................................. 135
B.1.2 Example File .......................................................................................................................... 135
B.2 MATLAB CODE ........................................................................................................................... 136
B.3 FILE FORMAT OF OUTPUT ............................................................................................................. 142

APPENDIX C MEASUREMENTS OF SELECT SHAPES OF THE DUAL-DUCTED FAN MODEL .............................................................................................................................. 143
LIST OF FIGURES

FIGURE 1-1: EXAMPLES OF DUCTED FANS CURRENTLY EMPLOYED TO PRODUCE VERTICAL THRUST: (A) LOCKHEED MARTIN F-35B LIGHTNING II (1), (b) HONEYWELL T-HAWK MICRO AIR VEHICLE (MAV) (2) ................................. 3
FIGURE 1-2: PIASECKI HELICOPTER AIRGEEP II ................................................................................................................. 3
FIGURE 1-3: PARLETT’S DUAL DUCTED FAN WIND TUNNEL MODEL (3) .............................................................................. 4
FIGURE 1-4: BELL/URBAN AERONAUTICS X-HAWK CONCEPT ......................................................................................... 4
FIGURE 1-5: URBAN AERONAUTICS (UA) AIRMULE .............................................................................................................. 5
FIGURE 1-6: STREAMLINE COMPARISON BETWEEN DUCTED (RING) AND UNDUCTED (WITHOUT RING) (6) ................ 6
FIGURE 1-7: CHANGE IN THRUST COEFFICIENT, C\textsubscript{T}, WITH RESPECT TO TIP CLEARANCES (7) ........................... 6
FIGURE 1-8: FLOW MODEL OF A SINGLE DUCTED FAN (4) ................................................................................................. 9
FIGURE 1-9: PROPELLER BLADE DESIGN (7) ....................................................................................................................... 9
FIGURE 1-10: DUCTED FAN DIMENSIONS (7) ................................................................................................................... 10
FIGURE 2-1: THREE-VIEW OF THE BASELINE MODEL SUB-STRUCTURE IN SOLIDWORKS ........................................... 15
FIGURE 2-2: SHAPING OF THE BASELINE DUAL FAN FUSELAGE ..................................................................................... 16
FIGURE 2-3: BASELINE DUAL FAN MODEL: (A) TOP VIEW, (B) SIDE VIEW, (C) ISOMETRIC VIEW .................................. 17
FIGURE 2-4: DEFINITION OF ROTOR INCIDENCE ANGLE .................................................................................................. 17
FIGURE 2-5: GEOMETRIC PROPERTIES FOR THE DUAL DUCTED FAN ROTORS: (A) CHORD DISTRIBUTION, (b) PITCH DISTRIBUTION, (c) MAX THICKNESS DISTRIBUTION.................................................................................................................. 19
FIGURE 2-6: E-FLITE POWER 32 BRUSHLESS OUT-RUNNER MOTOR ............................................................................... 20
FIGURE 2-7: SORENSEN DLM 32-95E POWER SUPPLY ......................................................................................................... 21
FIGURE 2-8: WIRING FROM A POWER SUPPLY TO A MOTOR ............................................................................................ 23
FIGURE 2-9: STREAMLINED NOSE ASSEMBLED OVER BASELINE MODEL (SOLIDWORKS 2010): (A) TOP VIEW, (B) SIDE VIEW ........................................................................................................................................ 24
FIGURE 2-10: FINAL DESIGN OF THE STREAMLINED NOSE: (A) TOP VIEW OF FIRST PART (LARGER SECTION), (b) ISOMETRIC VIEW OF FIRST PART, (c) TOP VIEW OF SECOND PART (SMALLER SECTION), (d) ISOMETRIC VIEW OF SECOND PART .................................................................................................................. 25
FIGURE 2-11: ASSEMBLED STREAMLINED MODEL: (A) TOP VIEW, (B) SIDE VIEW, (c) MOUNTED ON FORCE BALANCE IN APB WIND TUNNEL .................................................................................................................. 26
FIGURE 3-1: TESTO 416 MINI-VANE ANEMOMETER .............................................................................................................. 27
FIGURE 3-2: FIVE-HOLE PITOT PROBE .............................................................................................................................. 28
FIGURE 3-3: VALIDYNE DP103 PRESSURE TRANSDUCERS USED WITH FIVE-HOLE PITOT PROBE ................................. 29
FIGURE 3-4: VALIDYNE AMPLIFIERS USED WITH DP103 TRANSDUCERS FOR FIVE-HOLE PITOT PROBE ..................... 30
FIGURE 3-5: ROTATIONAL TRAVERSE SET-UP FOR CALIBRATION ...................................................................................... 31
FIGURE 3-6: FIVE-HOLE PITOT PROBE CALIBRATION SET-UP .......................................................................................... 31
FIGURE 3-7: SCREENSHOT OF LABVIEW PROGRAM USED FOR FIVE-HOLE PITOT PROBE CALIBRATION ................. 33
FIGURE 3-8: ENLARGED DAQ CONTROL TAB AND MOTOR CONTROL TAB OF LABVIEW PROGRAM ......................... 33
FIGURE 3-9: FIVE-HOLE PITOT PROBE YAW AND PITCH COEFFICIENT CALIBRATION ........................................ 35
FIGURE 3-10: TOTAL PRESSURE COEFFICIENT FOR CHANGES IN PROBE PITCH ANGLE .................................. 36
FIGURE 3-11: STATIC PRESSURE COEFFICIENT FOR CHANGES IN PROBE PITCH ANGLE ................................. 36
FIGURE 3-12: CLOSE-UP OF UNITED SENSORS KIEL PROBE ........................................................................... 37
FIGURE 3-13: KIEL PROBE FLOW ANGLE SENSITIVITY CALIBRATION SETUP .................................................. 39
FIGURE 3-14: KIEL PROBE FLOW ANGLE SENSITIVITY ................................................................................... 40
FIGURE 3-15: KIEL PROBE STATIC PRESSURE CALIBRATION ........................................................................... 41
FIGURE 3-16: SIDE VIEW OF HAMMOND WIND TUNNEL .................................................................................. 43
FIGURE 3-17: HAMMOND FORCE AND MOMENT PLATE BALANCE ................................................................. 44
FIGURE 3-18: TOP VIEW OF THE ACADEMIC PROJECTS BUILDING (APB) WIND TUNNEL .................................. 46
FIGURE 3-19: APB FORCE AND MOMENT PYRAMID BALANCE ........................................................................ 47
FIGURE 4-1: FLOW MEASUREMENT LOCATIONS WITH THE MINI-VANE ANEMOMETER .................................. 52
FIGURE 4-2: AXIAL VELOCITY ABOVE THE ROTOR ALONG THE BASELINE MODEL CENTERLINE MEASURED BY THE MINI-VANE ANEMOMETER AT 6000 RPM IN HOVER: (A) AFT FAN, (B) FORWARD FAN ................................. 53
FIGURE 4-3: AXIAL VELOCITY BELOW THE ROTOR ALONG THE BASELINE MODEL CENTERLINE MEASURED BY THE MINI-VANE ANEMOMETER AT 6000 RPM IN HOVER: (A) AFT FAN, (B) FORWARD FAN ........................................................................... 54
FIGURE 4-4: FIVE-HOLE PROBE AND KIEL PROBE LOCATIONS FOR FLOW MEASUREMENT WITHIN THE FORWARD DUCT ................................................................................................................ 55
FIGURE 4-5: MODEL COORDINATE DEFINITIONS FOR RESULTS FROM FIVE-HOLE PITOT PROBE .................. 57
FIGURE 4-6: VELOCITY MAGNITUDE AND COMPONENTS MEASURED BY THE FIVE-HOLE PITOT PROBE ABOVE THE FORWARD ROTOR OF THE BASELINE MODEL IN HOVER: (A) 4000 RPM AND (B) 6000 RPM ................................. 58
FIGURE 4-7: VELOCITY MAGNITUDE AND COMPONENTS MEASURED BY THE FIVE-HOLE PITOT PROBE BELOW THE FORWARD ROTOR OF THE BASELINE MODEL IN HOVER (A) 4000 RPM AND (B) 6000 RPM ........................................................................... 59
FIGURE 4-8: DOWNSTREAM FLOW ANGLE, $\theta$, MEASURED BY THE FIVE-HOLE PITOT PROBE WITHIN THE FORWARD FAN OF THE BASELINE MODEL OPERATING IN HOVER AS A FUNCTION OF AZIMUTHAL ANGLE: (A) ABOVE THE ROTOR, (B) BELOW THE ROTOR ........................................................................................................ 60
FIGURE 4-9: FLOW ANGLE, $\theta$, IN THE FORWARD FAN ALONG THE CENTERLINE OF THE BASELINE MODEL MEASURED BY THE FIVE-HOLE PROBE IN HOVER ........................................................................................................ 61
FIGURE 4-10: TOTAL THRUST (THE SUM OF ROTOR THRUST AND DUCT LIFT) FOR VARIOUS ROTOR RPMS MEASURED BY THE APB BALANCE WHILE OPERATING IN HOVER ......................................................... 62
FIGURE 4-11: THRUST COEFFICIENT FOR VARIOUS RPMS ............................................................................... 63
FIGURE 4-12: PITCHING MOMENT WHILE IN HOVER FOR VARIOUS RPMS ......................................................... 63
FIGURE 4-13: STREAMLINES FROM A SINGLE VERTICAL SMOKE WIRE PLACED AT THE MODEL CENTERLINE .................................................................................................................. 65
FIGURE 4-14: CLOSE-UP OF FLOW SEPARATION OVER THE FORWARD DUCT LIP ALONG THE CENTERLINE AT AN ANGLE OF ATTACK OF -10° AND ADVANCE RATIO OF 0.08 ........................................................................... 66
FIGURE 4-15: CLOSE-UP OF FLOW SEPARATION OVER THE FORWARD DUCT LIP ALONG THE CENTERLINE AT AN ANGLE OF ATTACK OF -10° AND ADVANCE RATIO OF 0.24 ........................................................................... 66
FIGURE 4-16: VELOCITY MAGNITUDES WITHIN THE FORWARD DUCT OF THE BASELINE MODEL MEASURED BY THE KIEL PROBE AND THE FIVE-HOLE PITOT PROBE: (a) ABOVE THE ROTOR AT $V_{WT}=6.1$ m/s, (b) ABOVE THE ROTOR AT $V_{WT}=10.7$ m/s, (c) BELOW THE ROTOR AT $V_{WT}=6.1$ m/s, (d) BELOW THE ROTOR AT $V_{WT}=10.7$ m/s ................................................................................................................................................ 68

FIGURE 4-17: AVERAGE OF VELOCITY MAGNITUDES MEASURED BY THE KIEL PROBE AND FIVE-HOLE PITOT PROBE WITHIN THE FORWARD DUCT OF THE BASELINE MODEL: (a) ABOVE THE ROTOR AT $V_{WT}=6.1$ m/s, (b) ABOVE THE ROTOR AT $V_{WT}=10.7$ m/s, (c) BELOW THE ROTOR AT $V_{WT}=6.1$ m/s, (d) BELOW THE ROTOR AT $V_{WT}=10.7$ m/s ................................................................................................................................................ 69

FIGURE 4-18: DOWNSTREAM FLOW ANGLE, $\theta$, WITHIN THE FORWARD FAN AS A FUNCTION OF AZIMUTHAL ANGLE AT AN ADVANCE RATIO OF 0.08: (a) ABOVE THE ROTOR, (b) BELOW THE ROTOR ................................................................................................. 71

FIGURE 4-19: DOWNSTREAM FLOW ANGLE WITHIN THE FORWARD FAN ALONG THE CENTERLINE OF THE MODEL AT AN ADVANCE RATIO OF 0.08 ................................................................................................................................. 72

FIGURE 4-20: DOWNSTREAM FLOW ANGLE, $\theta$, WITHIN THE FORWARD FAN AS A FUNCTION OF AZIMUTHAL ANGLE AT AN ADVANCE RATIO OF 0.21: (a) ABOVE THE ROTOR, (b) BELOW THE ROTOR ................................................................................................. 73

FIGURE 4-21: DOWNSTREAM FLOW ANGLE WITHIN THE FORWARD FAN ALONG THE CENTERLINE OF THE MODEL AT AN ADVANCE RATIO OF 0.21 ................................................................................................................................................ 73

FIGURE 4-22: DIMENSIONAL AND NON-DIMENSIONAL TOTAL LIFT AT 0° ANGLE OF ATTACK ........................................ 75

FIGURE 4-23: DIMENSIONAL AND NON-DIMENSIONAL TOTAL LIFT AT -5° ANGLE OF ATTACK ........................................ 76

FIGURE 4-24: DIMENSIONAL AND NON-DIMENSIONAL TOTAL LIFT AT -10° ANGLE OF ATTACK ........................................ 76

FIGURE 4-25: DIMENSIONAL AND NON-DIMENSIONAL NET DRAG AT 0° ANGLE OF ATTACK ........................................ 78

FIGURE 4-26: DIMENSIONAL AND NON-DIMENSIONAL NET DRAG AT -5° ANGLE OF ATTACK ........................................ 78

FIGURE 4-27: DIMENSIONAL AND NON-DIMENSIONAL NET DRAG AT -10° ANGLE OF ATTACK ........................................ 79

FIGURE 4-28: NON-DIMENSIONAL LIFT AND DRAG AT VARIOUS ANGLES OF ATTACK (OPEN SYMBOL = 6000 RPM, CLOSED SYMBOL = 8000 RPM) ........................................................................................................... 80

FIGURE 4-29: NET DRAG OF MODEL WHILE MAINTAINING CONSTANT LIFT BY CHANGING ANGLE OF ATTACK: (a) LIFT = 16.6 N, (b) LIFT = 26.2 N ................................................................................................................................................ 81

FIGURE 4-30: “RAM DRAG” RESULTING IN A NOSE-UP PITCHING MOMENT ........................................................................ 81

FIGURE 4-31: DIMENSIONAL AND NON-DIMENSIONAL PITCHING MOMENT AT 0° ANGLE OF ATTACK ........................................ 82

FIGURE 4-32: LIFT, DRAG, AND PITCHING MOMENT FOR VARIOUS ANGLES OF ATTACK AT 19.8 m/s: (a) 6000 RPM, (b) 8000 RPM ................................................................................................................................................ 83

FIGURE 5-1: CAD MODEL USED FOR CFD ......................................................................................................................... 85

FIGURE 5-2: MESH OF MODEL GENERATED AND PLACED INSIDE THE HAMMOND WIND TUNNEL TEST SECTION .......... 85

FIGURE 5-3: HYBRID BLOCK-STRUCTURED AND OVERSET MESH FOR THE MODELED ROTOR SIMULATIONS: (a) SURFACE MESH, (b) SLICE THROUGH THE HORIZONTAL PLANE, (c) SLICE THROUGH THE VERTICAL SYMMETRY PLANE [239 BLOCKS, 5.0M CELLS, 5.5M POINTS] ................................................................................................................................................ 87

FIGURE 5-4: PRESSURE AND VELOCITY CONTOURS AT 18.3 m/sec, 6000 RPM, AND 0° ANGLE OF ATTACK: (a) SURFACE STATIC PRESSURE, (b) DOWNSTREAM VELOCITY MAGNITUDE ................................................................................................................................................ 88

FIGURE 5-5: PARTICLE PATHS IN A FORWARD FLIGHT SIMULATION AT 18.3 m/sec AND 0° ANGLE OF ATTACK [FLOW IS LEFT TO RIGHT, RED RELEASED IN FORWARD DUCT, GREEN RELEASED IN AFT DUCT, BLACK RELEASED IN FUSELAGE WAKE] ................................................................................................................................................ 89
FIGURE 5-6: COMPARISON OF MOMENTUM SOURCE CFD FORCES TO EXPERIMENTAL FORCES: (A) TOTAL LIFT, (B) NET DRAG........................................................................................................... 90

FIGURE 5-7: HYBRID BLOCK-STRUCTURED AND OVERSET MESH FOR THE RESOLVED ROTOR SIMULATIONS: (A) SURFACE MESH, (B) SLICE THROUGH THE HORIZONTAL PLANE, (C) DETAIL OF AFT ROTOR BLADE SHOWING TIP GAP [309 BLOCKS, 12.4M CELLS, 13.5M POINTS] ................................................................. 91

FIGURE 5-8: SURFACE STATIC PRESSURE CONTOURS OF THE FULLY-RESOLVED SIMULATION AT 18.3 M/SEC FORWARD SPEED AT 6000 RPM: (A) TOP-FORWARD VIEW, (B) BOTTOM-FORWARD VIEW ............. 93

FIGURE 5-9: LIFT FORCE HISTORY FROM THE FULLY-RESOLVED ROTOR SIMULATION AT VWT=18.3 M/SEC, 6000 RPM, AND AN ANGLE OF ATTACK OF 0° [RED = FUSELAGE, GREEN = AFT ROTOR, BLUE = FORWARD ROTOR] .. 94

FIGURE 5-10: COMPARISON OF FULLY-RESOLVED CFD FORCES TO THE EXPERIMENTAL FORCE RESULTS: (A) TOTAL LIFT, (B) NET DRAG ............................................................................................................................ 94

FIGURE 6-1: STREAMLINES OVER THE STREAMLINED NOSE: (A) M =0.08, (B) M =0.12, (C) M =0.12 (FULL VIEW) ...... 98

FIGURE 6-2: VELOCITY MAGNITUDES WITHIN THE FORWARD DUCT OF THE STREAMLINED MODEL MEASURED BY THE KIEL PROBE AND THE FIVE-HOLE PITOT PROBE: (A) ABOVE THE ROTOR AT VWT=6.1 M/s, (B) ABOVE THE ROTOR AT VWT=10.7 M/s, (C) BELOW THE ROTOR AT VWT=6.1 M/s, (D) BELOW THE ROTOR AT VWT=10.7 M/s ........................................................................................................ 99

FIGURE 6-3: COMPARISON OF THE AVERAGED VELOCITY MAGNITUDES FROM THE KIEL PROBE AND THE FIVE-HOLE PITOT PROBE FOR THE BASELINE AND STREAMLINED MODEL ABOVE THE ROTOR AT 70% RADIUS: (A) VWT=6.1 M/SEC, (B) VWT=10.7 M/SEC................................................................. 100

FIGURE 6-4: COMPARISON OF THE AVERAGED VELOCITY MAGNITUDES FROM THE KIEL PROBE AND THE FIVE-HOLE PITOT PROBE FOR THE BASELINE AND STREAMLINED MODEL BELOW THE ROTOR AT 70% RADIUS: (A) VWT=6.1 M/SEC, (B) VWT=10.7 M/SEC................................................................. 102

FIGURE 6-5: DIMENSIONAL AND NON-DIMENSIONAL LIFT FOR THE STREAMLINED AND BASELINE MODEL AT 0° ANGLE OF ATTACK ........................................................................................................ 103

FIGURE 6-6: DIMENSIONAL AND NON-DIMENSIONAL LIFT FOR THE STREAMLINED AND BASELINE MODEL AT -5° ANGLE OF ATTACK ................................................................. 104

FIGURE 6-7: DIMENSIONAL AND NON-DIMENSIONAL LIFT FOR THE STREAMLINED AND BASELINE MODEL AT -10° ANGLE OF ATTACK ................................................................. 104

FIGURE 6-8: DIMENSIONAL AND NON-DIMENSIONAL DRAG FOR THE STREAMLINED AND BASELINE MODEL AT 0° ANGLE OF ATTACK ................................................................. 106

FIGURE 6-9: DIMENSIONAL AND NON-DIMENSIONAL DRAG FOR THE STREAMLINED AND BASELINE MODEL AT -5° ANGLE OF ATTACK ................................................................. 107

FIGURE 6-10: DIMENSIONAL AND NON-DIMENSIONAL DRAG FOR THE STREAMLINED AND BASELINE MODEL AT -10° ANGLE OF ATTACK ................................................................. 108

FIGURE 6-11: DIMENSIONAL AND NON-DIMENSIONAL PITCHING MOMENT FOR THE STREAMLINED AND BASELINE MODEL AT 0° ANGLE OF ATTACK ................................................................. 109

FIGURE 6-12: DIMENSIONAL AND NON-DIMENSIONAL PITCHING MOMENT FOR THE STREAMLINED AND BASELINE MODEL AT -5° ANGLE OF ATTACK ................................................................. 110

FIGURE 6-13: DIMENSIONAL AND NON-DIMENSIONAL PITCHING MOMENT FOR THE STREAMLINED AND BASELINE MODEL AT -10° ANGLE OF ATTACK ................................................................. 110

FIGURE 7-1: POSSIBLE STRAIN GAUGE LOCATIONS FOR MEASURING ROTOR THRUST ................................................................. 123
FIGURE 7-2: OPEN FORWARD DUCT CONCEPT

FIGURE 7-3: TOP VIEW OF BASELINE MODEL DISPLAYING POSSIBLE CUT LINES: (A) ANGLED CUT, (B) STRAIGHT CUT
LIST OF TABLES

TABLE 2-1: DUAL DUCTED FAN DESIGN PARAMETERS ................................................................. 15
TABLE 2-2: TIP GAP MEASUREMENTS OF THE DUAL-DUCTED FAN ROTORS ................................. 19
TABLE 2-3: PARTS LIST FROM POWER SUPPLIES TO MOTORS .............................................................. 22
TABLE 3-1: FIVE-HOLE PROBE PRESSURE TRANSDUCER CALIBRATION ......................................... 30
TABLE 3-2: HAMMOND BALANCE INFLUENCE COEFFICIENT MATRIX ........................................... 45
TABLE 3-3: APB FORCE AND MOMENT BALANCE INFLUENCE COEFFICIENT MATRIX ................... 50
TABLE 4-1: KIEL PROBE AND FIVE-HOLE PITOT PROBE MEASUREMENT LOCATION WITHIN THE FORWARD DUCT ................................................................. 56
TABLE 6-1: PERCENT DIFFERENCE IN DRAG BETWEEN THE STREAMLINED MODEL AND BASELINE MODEL AT A FUSELAGE ANGLE OF ATTACK OF 0° (NEGATIVE REFERS TO A DRAG DECREASE DUE TO THE STREAMLINED NOSE) ........................................ 106
TABLE 6-2: PERCENT DIFFERENCE IN DRAG BETWEEN THE STREAMLINED MODEL AND BASELINE MODEL AT A FUSELAGE ANGLE OF ATTACK OF -5° (NEGATIVE REFERS TO A DRAG DECREASE DUE TO THE STREAMLINED NOSE) .................................................................................. 107
TABLE 6-3: PERCENT DIFFERENCE IN DRAG BETWEEN THE STREAMLINED MODEL AND BASELINE MODEL AT A FUSELAGE ANGLE OF ATTACK OF -10° (NEGATIVE REFERS TO A DRAG DECREASE DUE TO THE STREAMLINED NOSE) .................................................................................. 108
# NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Length of Fuselage Centerbody</td>
</tr>
<tr>
<td>A</td>
<td>Ducted fan disk area (Sum of both ducts)</td>
</tr>
<tr>
<td>c</td>
<td>Duct Chord Length</td>
</tr>
<tr>
<td>$C_{D,TIP}$</td>
<td>Drag Coefficient Non-Dimensionalized by the Rotor Tip Speed</td>
</tr>
<tr>
<td>$C_{L,TIP}$</td>
<td>Lift Coefficient Non-Dimensionalized by the Rotor Tip Speed</td>
</tr>
<tr>
<td>$C_{m,TIP}$</td>
<td>Pitching Moment Coefficient Non-Dimensionalized by the Rotor Tip Speed</td>
</tr>
<tr>
<td>$C_P$</td>
<td>Pressure Coefficient</td>
</tr>
<tr>
<td>$C_{PP}$</td>
<td>Pitch Coefficient for the Five-Hole Pitot Probe</td>
</tr>
<tr>
<td>$C_{P,STATIC}$</td>
<td>Static Pressure Coefficient of the Five-Hole Pitot Probe</td>
</tr>
<tr>
<td>$C_{P,TOTAL}$</td>
<td>Static Pressure Coefficient of the Five-Hole Pitot Probe</td>
</tr>
<tr>
<td>$C_{PY}$</td>
<td>Yaw Coefficient for the Five-Hole Pitot Probe</td>
</tr>
<tr>
<td>$C_T$</td>
<td>Thrust Coefficient</td>
</tr>
<tr>
<td>D</td>
<td>Duct Diameter</td>
</tr>
</tbody>
</table>

\[
\begin{align*}
a &= \text{Length of Fuselage Centerbody} \\
A &= 2 \left( \pi \frac{D^2}{4} \right) \\
c &= \text{Duct Chord Length} \\
C_{D,TIP} &= \frac{\text{Net Drag}}{\frac{1}{2} \rho V_{Tip}^2 A} \\
C_{L,TIP} &= \frac{\text{Total Lift}}{\frac{1}{2} \rho V_{Tip}^2 A} \\
C_{m,TIP} &= \frac{\text{Pitching Moment}}{\frac{1}{2} \rho V_{Tip}^2 AR_m} \\
C_P &= \frac{P_T - P_{S,LOCAL}}{\frac{1}{2} \rho V_{LOCAL}^2} \\
C_{PP} &= \frac{P_4 - P_S}{P_1 - \frac{1}{4} (P_2 + P_3 + P_4 + P_5)} \\
C_{P,STATIC} &= \frac{\frac{1}{4} (P_2 + P_3 + P_4 + P_5) - P_S}{P_1 - \frac{1}{4} (P_2 + P_3 + P_4 + P_5)} \\
C_{P,TOTAL} &= \frac{P_1 - P_T}{P_1 - \frac{1}{4} (P_2 + P_3 + P_4 + P_5)} \\
C_{PY} &= \frac{P_2 - P_3}{P_1 - \frac{1}{4} (P_2 + P_3 + P_4 + P_5)} \\
C_T &= \frac{T}{\rho AV_{Tip}^2} \\
D &= \text{Duct Diameter}
\end{align*}
\]
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h_{\text{CENTERLINE}}$</td>
<td>Max Thickness of the Fuselage Centerbody along the Centerline of the Model</td>
</tr>
<tr>
<td>$h_{\text{MAX}}$</td>
<td>Overall Max Thickness of the Fuselage Centerbody</td>
</tr>
<tr>
<td>$P_{\text{KIEL(MAX/MIN)}}$</td>
<td>Max or Min Pressure Sensed by Kiel Probe</td>
</tr>
<tr>
<td>$P_s$</td>
<td>Ambient Static Pressure</td>
</tr>
<tr>
<td>$P_{s,\text{LOCAL}}$</td>
<td>Local Static Pressure within the Duct</td>
</tr>
<tr>
<td>$P_T$</td>
<td>Total Pressure</td>
</tr>
<tr>
<td>$R$</td>
<td>Rotor Radius</td>
</tr>
<tr>
<td>$R_m$</td>
<td>Distance from Balance Mounting Point (Longitudinal Center of Centerbody) on Model to Front Hub</td>
</tr>
<tr>
<td>$t$</td>
<td>Duct Thickness</td>
</tr>
<tr>
<td>$T$</td>
<td>Total Thrust</td>
</tr>
<tr>
<td>$V_i$</td>
<td>Rotor Induced Velocity</td>
</tr>
<tr>
<td>$V_{\text{LOCAL}}$</td>
<td>Local Velocity Magnitude within the Duct</td>
</tr>
<tr>
<td>$V_{\text{TIP}}$</td>
<td>Rotor Tip Speed</td>
</tr>
<tr>
<td>$V_{\text{WT}}$</td>
<td>Wind Tunnel Velocity</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Angle of Attack</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Rotor Incidence Angle</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Downstream Flow Angle</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Ambient Air Density</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Advance Ratio $\frac{V_{\text{WT}}}{V_{\text{TIP}}}$</td>
</tr>
<tr>
<td>$\psi$</td>
<td>Azimuthal Angle (deg)</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

First and foremost I would like to thank my entire family, especially Mom, Dad, and Jered, as well as my friends and coworkers. Their continued support and guidance always kept me going when times were rough. They were there to push me to achieve beyond my goals. Without them, I wouldn’t be the quality of person I am today.

Just as important is the one that will always have a special place in my heart, Lauren. Her encouragement and support has gotten me through times when I was overwhelmed with responsibilities. She put up with my late night antics finishing reports and made sure I continued working hard instead of slacking off. I am forever grateful that she stuck with me through these times.

I would like to acknowledge and extend much thanks to my advisor, Dr. Dennis McLaughlin. I cannot thank him enough for all he has done for me throughout my undergraduate and graduate career. To be able to participate in the ducted fan program at Penn State was a great experience. The knowledge he has passed on to me has been invaluable.

I would like to thank the sponsors of this project, the Office of Naval Research for initially funding the project and the Defense Research Related Unit Program offered through the Penn State Office of the Senior Vice President for Research who provided further support. Without their financial contributions, this project would not be possible.

In addition, I would like to thank all that have contributed their assistance and knowledge to the ducted fan project: Dr. Naipei (Peter) Bi, Dr. James Dreyer, Dr. Edward Smith, Dr. Robert Kunz, Leighton Myers, Jason Halwick, Robert Thacker-Dey, Nidhi Sikarwar, Nate Morgan, Greg Davis, Nick Hoburn, Russell Powers, Ryan Stanley, and Nate Dependbusch. It was a pleasure being able to work with all of you.
Chapter 1

INTRODUCTION

The idea of a vehicle that could lift itself vertically from the ground and hover motionless in the air was probably born at the same time that man first dreamed of flying.

- Igor Ivanovitch Sikorsky (1955)

1.1 BACKGROUND AND MOTIVATION

The freedom and flexibility of a vehicle capable of vertical takeoff and landing (VTOL) has been sought after since Leonardo da Vinci first dreamed up the Air Screw. Such a vehicle wouldn’t be restricted by the lack of a runway and can hover in place when necessary. Helicopters have primarily fulfilled this role. Other designs, such as tail-sitters or tilt-wings, have been attempted, but are typically much less efficient than helicopter because of the trade-offs required. One alternative design that holds promise but had seen little field testing until recently are ducted fan vehicles. Due to the military’s desire for small unmanned aerial vehicles (UAVs) to provide increased awareness in the battlefield, ducted fans have quickly gained recognition in the UAV class.

Many of the current military UAVs are fixed-wing aircraft, which not only lack the ability to hover but have landing and takeoff restrictions as well. Those operated remotely from a base still require the runway space and time allotments for takeoff and landing. The launch and recovery of the smaller fixed-wing UAVs being operated from the field require specialized techniques that may limit the mobility or readiness. Ducted fans allow for vertical takeoffs and
landings with only the restriction of having a clearing slightly larger than the footprint of the vehicle.

In the 1930’s, Luigi Stipa and Ludwig Kort developed the ducted propeller, a propeller fitted within a non-rotating nozzle. Initially, this concept was used in marine applications. In the 1940’s, the ducted propeller evolved into the ducted fan as the turbofan, providing thrust for fixed-wing aircraft. Typically, the difference between a ducted propeller and ducted fan is the solidity of the rotor. A fan is normally much higher in solidity. However, the terms fan and propeller have often been used interchangeably when referring to them being ducted, much like the use of duct and shroud.

Ducted fans have since been utilized in helicopters as anti-torque devices. Originally conceived by Sud Aviation in the 1960s, the ducted fan replaced the conventional open tail rotor in some helicopter models by enclosing a fan in the vertical tail fin. This anti-torque design is also known a fan-in-fin, fantail, or fenestron design and can be found in the RAH-66 Comanche and many Eurocopter designs today.

The focus has now shifted from using ducted fans as a source of thrust to a source of lift. Ducted fans are being incorporated in manned and unmanned vertical takeoff and landing (VTOL) vehicles in order to provide vertical thrust, as well as propulsion in some cases. The short takeoff vertical landing (STOVL) F-35B Lightning II, Figure 1-1 (a), is a prime example of the current use of ducted fans being used for lift in production. It uses a ducted fan just behind the cockpit coupled with the engine nozzle to produce the lift needed for vertical takeoff and landing. Honeywell’s T-Hawk, shown in Figure 1-1 (b), is a single ducted fan micro air vehicle (MAV) that relies on the ducted fan for vertical thrust and propulsion. The model tilts toward the direction of travel in order to gain a component of thrust for propulsion. Directional vanes lie at the exit of the duct and are capable of controlling the vehicle by directing the outflow.
Figure 1-1: Examples of Ducted Fans Currently Employed to Produce Vertical Thrust: (a) Lockheed Martin F-35B Lightning II (1), (b) Honeywell T-Hawk Micro Air Vehicle (MAV) (2)

While the F-35B and single ducted fan UAVs are on the forefront of the expansion of ducted fan use, the focus of the work presented in this thesis is on a concept vehicle more unique. Referred to as a dual-ducted fan vehicle, this conceptual vehicle is equipped with two ducted fans whose main purpose is to produce vertical thrust. This type of vehicle was first attempted by Piasecki Helicopter when they were contracted by the US Army to develop a flying jeep in 1957, Figure 1-2.

Figure 1-2: Piasecki Helicopter AirGeep II
In 1960, Parlett and NASA published technical reports regarding wind tunnel tests on a
dual-ducted fan model shown in Figure 1-3.\textsuperscript{(3)} Little research specifically on dual-ducted fan
vehicles was made available to the public after Parlett’s work. However, two conceptual dual-
ducted fan vehicles have recently emerged with one now entering tethered flight tests. The first
conceptual vehicle, the X-Hawk shown in Figure 1-4, was a joint effort design between Bell
Helicopter and Urban Aeronautics. Urban Aeronautics then continued the design effort and are
currently flight testing the AirMule shown in Figure 1-5.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{Figure1-3.png}
\caption{Parlett’s Dual Ducted Fan Wind Tunnel Model\textsuperscript{(3)}}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{Figure1-4.png}
\caption{Bell/Urban Aeronautics X-Hawk Concept}
\end{figure}
1.2 DUCTED FAN ADVANTAGES

Placing a duct or shroud around a propeller or fan creates several aerodynamic advantages. When comparing a ducted fan to an open propeller at the same power loading, the ducted fan can be up to 30% more efficient. This allows for a ducted fan with the same area as an isolated propeller to either have a lower power required or produce more thrust. The other advantage would be a reduction in area. This efficiency gain is made possible because of the duct in several ways. One is the ability to control the fan’s wake, or outflow, and increase its effective area.

The design of the duct is crucial, as it will determine the performance gains measured over an isolated rotor. The duct itself will produce a thrust force when in a static hover. The flow being accelerated around the upper duct radii will result in a low pressure, while a higher pressure will be experienced on the lower edges of the duct. The thicker duct will take better advantage of this pressure differential and produce more thrust. The duct can also be designed so that the wake won’t contract like it would with an isolated propeller. By decreasing the wake contraction, the
mass flow rate is increased, therefore producing more thrust. (5) Figure 1-6 shows the possible
difference in streamlines between an open propeller and a ducted propeller.

![Streamline Comparison between Ducted (ring) and Unducted (without ring)](image)

Figure 1-6: Streamline Comparison between Ducted (ring) and Unducted (without ring) (6)

The effective area, or disk area that is producing thrust, is increased by placing a duct
around the rotor by reducing losses associated with the tip vortices. Research shows that if the
distance between the blade tips and the duct is less than one percent of the radius, thrust is
improved greatly, as shown by the thrust coefficients for an isolated rotor and three tip gap
distances of a ducted rotor in Figure 1-7 where thrust coefficient is a function of rotor speed. (7)
The duct interacts with the tip vortices and, depending on the tip gap, allows less of the vortices
to be ingested by the rotor.

![Change in Thrust Coefficient, \(C_T\), with respect to Tip Clearances](image)

Figure 1-7: Change in Thrust Coefficient, \(C_T\), with respect to Tip Clearances (7)
There is also a potential for the duct to reduce the noise radiated by the vehicle. If treated properly, the duct could absorb the noise produced by the rotor. This option would allow the ducted rotor to exhibit a smaller acoustic footprint when compared to an isolated rotor.

There are also some mechanical advantages of a ducted fan over an open propeller. First, because of the possible size reduction of the fan, a smaller aircraft can be built, like the X-Hawk or the MAVs. The duct also provides extra safety for personnel and for the aircraft. The duct helps shield personnel on the ground from the danger of the rotating blades, or if a blade were to catastrophically fail and come apart, the duct should contain the debris. It also protects the blades from objects that may strike them at low altitudes. This increases the flexibility of choosing a suitable landing location for a ducted fan aircraft. It has the advantage of fitting into tight places where a helicopter would have to avoid because of the open rotor, such as trees or electrical wires.

Complexity is a key factor in cost because of maintenance. The ducted fan can help reduce these costs because of its ability to eliminate the need for hinges and bearings for cyclic control. Instead, control vanes can be integrated into the flow, as they were on the T-Hawk in Figure 1-1(b) and the AirMule in Figure 1-5. These control vanes can be used maneuver the vehicle in all six degrees of freedom.

1.3 DISADVANTAGES

With all these advantages, why aren’t ducted fans used everywhere? Unfortunately, there are disadvantages to ducted fans that eliminate performance gains from the advantages discussed. Currently, the largest disadvantage of the ducted fan is its inefficiency in forward flight. Since the duct on a lifting is required to operate in edgewise flow, the ducted will create a large amount of drag in forward flight. The duct can be optimized for high-speed
forward flight, but this will come at the cost of reducing the efficiency at hover and low-speed flight.\textsuperscript{(8)} Even with optimization of the duct, the fuselage still remains a fairly large body in the flow and will result in parasite and pressure drag.

Adding a duct to an isolated rotor also results in additional weight. However, any isolated rotor must also have a fuselage attached. If the centerbody, body between the two rotors, of the dual-ducted fan is design to carry all the systems and addition payload, the weight gains due to the ducts can be offset or negated.

### 1.4 Blade and Duct Design

Marc de Piolenc and Wright state that for an efficient ducted fan, the design should primarily focus on the duct design and only secondarily on the rotor design.\textsuperscript{(5)} If the ducted fan is designed for mass flow augmentation, the rotor loading is mild resulting in a rotor design that is more flexible and forgiving. However, the design of the duct is much more important in achieving optimization, especially the inlet design.

The inlet of the duct is an important aspect of the duct design. It should resemble a converging nozzle that gradually accelerates the flow into the fan. Part of the flow’s static pressure will be converted to velocity, while the total pressure should remain unchanged.\textsuperscript{(5)} The lip radius of the duct inlet must be shaped carefully to avoid separation of flow when traveling edge-wise. If flow were to separate at the inlet, the effectiveness will drop greatly due to a loss in thrust\textsuperscript{(4)}. Separation is very important to avoid, however, any other changes to the inlet will not result in increased performance if operating at high thrust coefficients due to the large suction force created by the fan in these certain conditions\textsuperscript{(8)}.

The outlet is designed much like a diverging nozzle. This will cause the slipstream flow area to become larger and increase the efficiency, as mentioned earlier. The slipstream will not
converge as with the open propeller and, if designed correctly, will actually diverge creating a larger effective area. Care must be taken, however, that the slipstream isn’t over-expanded resulting in separation of the flow from the duct wall. This would result in unwanted pressure losses and would decrease efficiency. Research has shown that the duct exit area is the most dominant design variable with respect to performance \(^{(9)}\). The internal duct configuration can be seen in Figure 1-8.

![Figure 1-8: Flow Model of a Single Ducted Fan \(^{(4)}\)](image)

Blade design for a ducted rotor will be different than the blade designed for use on the main or tail rotor of a helicopter because of the different flow each experiences. The blade of a ducted rotor will not have the taper that would be seen on a helicopter blade because the inflow will ideally be more uniform across the rotor disk inside the duct. Figure 1-9 is an example of a blade design that could be utilized in a ducted fan.

![Figure 1-9: Propeller Blade Design \(^{(7)}\)](image)
The placement of the rotor inside the duct has also been a point of study. Performance varies based on the depth of the rotor at the fore, aft, and middle position of the duct. The rotor can be mounted from the end of the inlet radius, $z=0$, to a quarter of the inside diameter, $D$, of the duct without experiencing performance losses, refer to Figure 1-10\(^{(7)}\). If the rotor is placed at the fore of the duct within the varying radius of the inlet, it will not experience the uniform flow that it would deeper into the duct. If the fan is placed at the aft of the duct, the opportunity to use a variable exit area or thrust vectoring is lost.\(^{(5)}\)

---

**Figure 1-10: Ducted Fan Dimensions\(^{(7)}\)**
1.5 Objectives

The goal of this research is to further the understanding of the aerodynamics and operational constraints of a dual-ducted fan vehicle. In order to accomplish this goal, the objectives of the present work are to:

1. Design and fabricate a dual-ducted fan vehicle. The vehicle will be generic in shape so as to best support any future designs. It will be strong so as to withstand wind tunnel testing. It will also be a powered model so the two fans within the ducts can be operational.

2. Thoroughly examine the generic, baseline model in hover and forward flight. Flow measurements within the two ducts will be measured in hover. Total thrust and pitching moment will be measured on a force balance. In forward flight, flow visualization will be performed and flow measurements will be obtained within the forward duct. Force and moment measurements will be made via a force and moment balance.

3. Working with Dr. James Dreyer and the Penn State Applied Research Lab, two types of computational fluid dynamics simulation for a dual-ducted fan vehicle will be developed and validated using experimental measurements.

4. Modifications to the baseline dual-ducted fan wind tunnel model will be made in an effort to reduce the high drag associated with ducted fan vehicle and improve overall performance.
1.6 Thesis Overview

This thesis is organized into the following remaining chapters:

Chapter 2: Description of Wind Tunnel Models

A full explanation of the design and build phase of the dual-ducted fan wind tunnel model is contained in this chapter. The electronics used to operate the model are also presented. The design and build phase of a streamlined nose is also included.

Chapter 3: Experimental Techniques and Facilities

Details of the equipment used throughout the research are presented in this chapter and the processes followed in their used. The wind tunnel facilities provided by Penn State are also described.

Chapter 4: Baseline Model Results and Discussion

Flow measurements and force and moment measurements were obtained in hover for the baseline model. Flow visualization, flow measurements, and force and moment measurements were also obtained in forward flight. These results are presented and discussed in this chapter.
Chapter 5: Baseline Dual-Ducted Fan Computation Fluid Dynamics

The work of Dr. James Dreyer and the Applied Research Lab of Penn State on two types of computational modeling simulations for dual-ducted fan vehicles is summarized in this chapter.

Chapter 6: Comparisons of the Streamlined Model to the Baseline Model

A streamlined nose was designed, fabricated, and installed on the baseline model in an attempt to improve vehicle performance. Flow visualization, flow measurements, and force and moment measurements were obtained and are compared to those from the baseline model in this chapter.

Chapter 7: Conclusions and Suggestions for Future Work

The final chapter summarizes the conclusions of the current work on the dual-ducted fan and suggests some future work to be performed on the model.

Appendices

The appendices contain the codes used for calibrating the five-hole pitot probe and calculating an unknown flow. Measurements made along specific sections of the fuselage are also presented.
Chapter 2

DESCRIPTION OF WIND TUNNEL MODELS

2.1 DUAL FAN MODEL

2.1.1 Fuselage / Duct Design

During the initial design phase of the baseline dual ducted fan wind tunnel model, it was decided that the design should include parameters from other previously built models while remaining a very generic research test vehicle. The wind tunnel model was designed to represent a generic UAV aerial medevac unit scaled down to 1/10th the size. Table 2-1 displays design parameters from other dual ducted fan models that were available to the public and how the Penn State dual ducted fan model compares. These parameters include the length of the center-body (shortest distance from the tip of the forward rotor to the tip of the aft rotor), $a$, the duct chord, $c$, the maximum thickness of the center-body along the centerline of the model, $h_{\text{CENTERLINE}}$, the overall maximum thickness of the center-body, $h_{\text{MAX}}$, the max duct thickness, $t$, the placement of the rotors within the duct, $\text{Rotor Depth}$, possible inclination angles of the rotors, and approximate cargo capacity. Length measurements have been non-dimensionalized by the use of the duct diameter, $D$, or duct chord, $c$, and the approximate cargo capacity of the Penn State model is listed for the full-scale UAV medevac.
Table 2-1: Dual Ducted Fan Design Parameters

<table>
<thead>
<tr>
<th>Name</th>
<th>a/D</th>
<th>c/D</th>
<th>h_{CENTERLINE}/D</th>
<th>h_{MAX}/D</th>
<th>Duct t/c</th>
<th>Rotor Depth/c</th>
<th>Angle of Rotor Inclination</th>
<th>Approximate Cargo Capacity [m²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>PSU A (UAV)</td>
<td>0.65</td>
<td>0.40</td>
<td>0.60</td>
<td>0.65</td>
<td>0.40</td>
<td>0.53</td>
<td>Capable of ± 10 deg</td>
<td>2.37</td>
</tr>
<tr>
<td>Airgeep II</td>
<td>0.86</td>
<td>0.22</td>
<td>0.18</td>
<td>0.55</td>
<td>0.10</td>
<td>N/A</td>
<td>N/A</td>
<td>4.89</td>
</tr>
<tr>
<td>NASA Parlett</td>
<td>0.76</td>
<td>0.32</td>
<td>0.24</td>
<td>0.24</td>
<td>0.44</td>
<td>0.09</td>
<td>30.00 deg</td>
<td>1.04</td>
</tr>
<tr>
<td>UA Airmule</td>
<td>0.65</td>
<td>0.27</td>
<td>0.63</td>
<td>0.63</td>
<td>0.41</td>
<td>N/A</td>
<td>N/A</td>
<td>2.37</td>
</tr>
<tr>
<td>UA/Bell X-Hawk</td>
<td>0.95</td>
<td>0.28</td>
<td>0.61</td>
<td>0.94</td>
<td>0.33</td>
<td>0.40</td>
<td>N/A</td>
<td>11.44</td>
</tr>
</tbody>
</table>

The baseline dual-ducted fan wind tunnel model was designed and fabricated by Depenbusch as partial fulfillment of an undergraduate honor’s thesis. If more information pertaining to the design and fabrication of the dual-ducted fan is desired than what is presented in this thesis, please refer to Reference (10). The sub-structure, seen in Figure 2-1, consists of a steel frame, two PVC rings to shape the ducts, and two ¼-20 threaded rods running horizontally across the structure to provide hard-points for mounting to a force balance.

Figure 2-1: Three-View of the Baseline Model Sub-Structure in SolidWorks
The fuselage was constructed out of FOAMULAR 250 polystyrene foam insulation. This allowed for hand-shaping of the fuselage. Material could be removed through sanding or carving and material could be added by means of body putty. Once a shape had been decided upon, layers of fiberglass reinforced epoxy resin were laid over the fuselage. This would not only improve the appearance of the model but also improve the smoothness and strength qualities. The model can be seen with the naked foam insulation and body putty build-up in Figure 2-2, while Figure 2-3 displays several views of the finished model with the fiberglass treatment.

Figure 2-2: Shaping of the Baseline Dual Fan Fuselage
Built into the structure of the ducted fan was the ability to change the forward and rear rotor incidence angle independently. This was done by fabricating motor cradles in between the two longitudinal support beams. A pivot point was created at the top of the motor cradle while a slotted arc was cut at the bottom of the support beams. The motor cradle was tapped so once the rotor incidence angle is set, the pivot point bolt and the bolt within the slot are then tightened. The rotor incidence angle, or $\beta$ as defined in Figure 2-4, can be varied $\pm 10$ degrees. For the data presented in this thesis, $\beta$ remained at 0 degrees for all cases.
2.1.2 Rotors

Trade studies performed by Myers had determined that a good rotor that was available commercially off the shelf for the single ducted fan was a 3-bladed Master Airscrew rotor, 10 inches in diameter with 5 inches of pitch. However, more thrust was desired from the rotors to be used on the dual ducted fan model. A pusher and a tractor propeller would also have to be available so counter-rotating propellers could be used to balance the torque on the model.

The rotors decided upon for the Penn State dual-ducted fan model were Master Airscrew’s 3-bladed, 10-inch diameter propellers with 7 inches of pitch. The tips of the propellers were then trimmed and made square to fit inside the ducts of the model. White-out was placed on the lower surface of the rotor to serve as a white strip for the optical sensor that would be used for measuring rotor revolution speed. Using a prop balancer, the two propellers were balanced by using sandpaper on the rotor to remove weight or white-out on the tips of the rotor to add weight.

Measurements of the finalized rotor shapes were taken with the use of a precision dial caliper and a protractor. Chord, blade pitch, the maximum thickness, and the location of max thickness from the leading edge were measured close to the root, at mid-span, and near the tip for each rotor. These dimensions are displayed in Figure 2-5 for the forward and aft rotor as a function of radial location. As expected, the tractor and pusher rotors are almost identical.

The tip gap, or distance between the rotor tip and the duct wall, was also measured. Data from Sato shown in Figure 1-7 exemplifies the effect of the rotor tip gap on the performance. With the use of a feeler gauge, the tip gap of both rotors was measured and is presented in Table 2-2. Measurements made before and after the rotors were modified in order to balance them. The nominal tip gap was 2% of the rotor radius.
Figure 2-5: Geometric properties for the dual ducted fan rotors: (a) Chord Distribution, (b) Pitch Distribution, (c) Max Thickness Distribution.

Table 2-2: Tip Gap Measurements of the Dual-Ducted Fan Rotors

<table>
<thead>
<tr>
<th>Blade</th>
<th>Tip Clearance</th>
<th>% of Radius</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Initial [in.]</td>
<td>Final [in.]</td>
</tr>
<tr>
<td>Aft 1</td>
<td>0.076</td>
<td>0.087</td>
</tr>
<tr>
<td>Aft 2</td>
<td>0.096</td>
<td>0.095</td>
</tr>
<tr>
<td>Aft 3</td>
<td>0.079</td>
<td>0.095</td>
</tr>
<tr>
<td>Forward 1</td>
<td>0.091</td>
<td>0.091</td>
</tr>
<tr>
<td>Forward 2</td>
<td>0.082</td>
<td>0.082</td>
</tr>
<tr>
<td>Forward 3</td>
<td>0.097</td>
<td>0.097</td>
</tr>
</tbody>
</table>
2.1.3 Motor and Electronics

Initially, the model was designed and fabricated to contain E-flight Park 370 brushless out-runner motors, the same motor used by Leighton Myers on the 10-1 and 10-2 models. However, these motors did not meet the necessary power requirements to turn the higher pitched rotors at high speeds. It was opted to upgrade the motors to the E-flight Power 32 brushless out-runner motors, as seen in Figure 2-6. These motors were capable of turning the rotors at 10,000 RPM; however, larger motor cradles had to be fabricated in order to mount them.

![E-flite Power 32 Brushless Out-Runner Motor](image)

Instead of using batteries and having the concern of battery life during long tests, two Sorensen DLM 32-95E power supplies, seen in Figure 2-7, were purchased to power the two motors. They require a three-phase, four-wire 208 VAC source for power. They are capable of outputting 0-32 VDC and 0-95 amps. Depending on the application, they can be programmed to hold voltage steady while amperage is increased, or hold the amps steady while voltage is increased. These motors require a steady voltage while increasing amperage, which is the default setting of the power supplies. The voltage is set at 16.5 VDC for each motor, which is just below the maximum voltage of motors, 16.8. As higher rotor RPMs are desired, the amperage is increased by an Astro Servo Tester to be discussed in the section.
Table 2-3 and Figure 2-8 display the necessary wiring and electronics between the Sorensen power supply and an E-flite motor. It is important to point out several of the electronic items shown here. Following the wiring out of the back of the power supply, the first electronic component is the Eagle Tree Systems E-Logger. The E-Logger is a commercial data logging product capable of recording performance and tracking data of remote controlled aircraft. Numerous sensors are made available through Eagle Tree. In this setup, the only sensors used are the optical RPM sensor and the temperature sensor. The optical RPM sensor is placed just under the rotor, in line with the white-out strips near the root of the blades. A female Bayonet Neill-Concelman (BNC) connector is also spliced into this wiring so to provide coaxial cable access. This allows the RPM to also be tracked in LabView or a digital signal analyzer (DSA). The temperature sensor is placed at the base of the motor and inserted slightly inside. This will keep track of the motor temperature to make sure the motor doesn’t overheat. Also attached to the E-Logger is a USB link. This allows for real-time display of the measurements on a PC with the use of Eagle Tree’s proprietary software. The software also allows for post-processing of the measurements by supporting the use of tables and graphs.

The electronic speed control (ESC), a Castle Creations Phoenix-125, is next within the wiring. The ESC is responsible for managing the speed of the motor, acting as a breaking mechanism for the motor, and automatically cutting off power to the motor if sufficient voltage is not available to avoid damaging equipment. By connecting the Astro Servo Tester to the receiver
port on the ESC, the motor speed can now be controlled by servo tester rather than an expensive wireless hand-held radio. The servo tester is capable of changing the current pulse width supplied to the motor, which can increase or decrease the motor speed. If the power being supplied to the motor is $13 \ V_{DC}$ or less, the ESC will supply the servo tester power. Since the Sorensen power supplies are outputting $16.5 \ V_{DC}$, the servo testers would be overpowered. Therefore two more, much smaller, power supplies are used to power the two servo testers. These are set to output $5 \ V_{DC}$.

Table 2-3: Parts List from Power Supplies to Motors

<table>
<thead>
<tr>
<th>Item #</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Fork Terminal</td>
</tr>
<tr>
<td>2</td>
<td>Fuse Holder</td>
</tr>
<tr>
<td>3</td>
<td>60 Amp Fuse</td>
</tr>
<tr>
<td>4</td>
<td>Dean’s Plug</td>
</tr>
<tr>
<td>5</td>
<td>BNC Connector</td>
</tr>
<tr>
<td>6</td>
<td>Optical RPM Sensor</td>
</tr>
<tr>
<td>7</td>
<td>Temperature Sensor</td>
</tr>
<tr>
<td>8</td>
<td>USB to PC</td>
</tr>
<tr>
<td>9</td>
<td>Electronic Speed Control</td>
</tr>
<tr>
<td>10</td>
<td>Servo Tester</td>
</tr>
<tr>
<td>11</td>
<td>3-pin Connector</td>
</tr>
<tr>
<td>12</td>
<td>Banana Plug</td>
</tr>
<tr>
<td>13</td>
<td>12-AWG Wire in Red, Blue, and Black</td>
</tr>
<tr>
<td>14</td>
<td>E-Logger</td>
</tr>
<tr>
<td>15</td>
<td>Servo Wire</td>
</tr>
<tr>
<td>16</td>
<td>3-pin Servo Connector</td>
</tr>
</tbody>
</table>
2.2 STREAMLINED NOSE

2.2.1 Design and Fabrication

In order to gain a better understanding about the operation of the dual ducted fan model in edgewise flight and attempt to improve its performance, a more aerodynamic (streamlined) nose was designed, fabricated, and tested on the baseline model. The CAD file of the baseline model design was imported into SolidWorks 2010. With the assistance of Thacker-Dey, the streamlined nose was designed to fit over-top of the existing baseline blunt nose and attach using flat head bolts and embedded threaded inserts. A top and side view of the completed design can be seen in Figure 2-9. For an in-depth explanation of the design of the streamlined nose within SolidWorks, refer to Thacker-Dey’s undergraduate thesis. (12)
After the design had been finalized, it was saved as a .stl file and was sent to the Penn State Learning Factory for production in the Stratasys FDM-2000 rapid-prototype machine. It was found that the streamlined nose extended past the machines limits, which is 10x10x10 inches. The nose design would have to be modified in order to allow for rapid-prototyping. The decision was made to spilt the part in two pieces off the axis of symmetry, as shown in Figure 2-10. Material was added to the interior shell of the nose to increase area for attachment.
2.2.2 Attachment

After production, it was found that the nose fit was very tight in the width and loose in the height. Instead of stretching the nose across the width of the model and running the risk of over-stressing and breaking it, the width of the nose was increased by placing a 1/4-inch thick piece of lexan between the two pieces. Epoxy was then applied, along with two nuts and bolts for increased strength. In order to create a tight fit in the vertical direction, 1/8-inch thick and 1/4-inch wide foam strips with adhesive backing were applied to the nose until the desired fit was found.

Figure 2-11 displays several views of the streamlined nose attached to the baseline model. Attachment was achieved with the use of knife-threaded inserts with an internal thread for a 10-32 bolt and undercut flat head 10-32 bolts. Six 0.196 holes were drilled and counter-
sunk at 82 degrees for the flat heat bolts to mount flush to the nose. Six 9/32-inch holes were drilled in the model for the threaded-inserts. These were then threaded into the PVC ring, when possible, or epoxy was used to affix the threaded insert to the polystyrene foam insulation as best as possible. The attachment bolt locations can be seen in Figure 2-11(c).

Figure 2-11: Assembled Streamlined Model: (a) Top View, (b) Side View, (c) Mounted on Force Balance in APB Wind Tunnel
Chapter 3

EXPERIMENTAL TECHNIQUES AND FACILITIES

3.1 MINI-VANE ANEMOMETER

A hand-held Testo 416 Mini-Vane Anemometer, shown in Figure 3-1, was the first instrument used to quantify flow velocities through the two ducted fans. This instrument consists of a shrouded 5/8-inch vane anemometer on a telescoping shaft and a hand-held recorder. Within the shroud is a magnet which tracks blade passage of the vane anemometer. The hand-held recorder then processes this and displays the velocity in units of meters per second. The recorder can be used to display the real-time sensed velocity or an average velocity. Data displayed in this thesis used the averaging function of the recorder by pressing the ‘Mean’ button twice. The ‘Hold’ button is then pressed to start taking measurements at a rate of two per second. Real-time velocities will be shown until the ‘Mean’ button is pressed again, which stops the timer and displays the time-averaged velocity. For this thesis, velocities were averaged over a time period of 10 seconds. Testo claims the 416 anemometer is capable of sensing velocities from 0.6 m/sec (2.0 ft/sec) to 40 m/sec (131 ft/sec) and is accurate to ±0.2 m/sec (±0.7 ft/sec).

Figure 3-1: Testo 416 Mini-Vane Anemometer
3.2 FIVE-HOLE PITOT PROBE

3.2.1 Description

Another instrument used in the process of quantifying flow within the duct was a five-hole pitot probe. As the name implies and as seen in Figure 3-2, this instrument had five pressure ports in a pyramidal formation at the tip of a 90° pitot probe. This design allows for a pitot probe that is capable of measuring velocity components in three directions. Therefore, it was able to resolve velocity magnitude and flow angles in two planes perpendicular to each other within pitch and yaw angles of ±35° to the flow direction. If higher angles of flow are attempted to be measured, flow separation off of the probe will most likely occur and create error.

Each pressure port is connected via tygon tubing to five individual Validyne DP103 pressure transducers, pictured in Figure 3-3. The center port (1) is connected to transducer 1, which contains a #12 diaphragm and is rated to a maximum pressure of 0.14 kPa. The outer four ports are connected to their respective transducers and contain #10 diaphragms capable of measuring pressures up to 0.086 kPa. For more information on the five-hole pitot probe than this thesis provides, refer to Remaly’s thesis. (13)
3.2.2 Calibration

Calibration of the Validyne pressure transducers was accomplished by using tygon tubing to connect the five positive sides of the transducers together, as well as the total pressure port of a pitot-static probe and a digital manometer. This allowed for simultaneous calibration of all five transducers. The pitot-static probe was placed within the center of the flow of the Hammond axial jet, located in the basement of Hammond Building in rooms 6 and 8. The axial jet velocity was increased until the total pressure reached the maximum allowable pressure of the smallest diaphragm, 0.087 kPa. The gains were then set on the amplifiers, Figure 3-4, so that all the transducers read approximately eight volts. The pressure was then relieved by turning off the jet and a zero was taken. Incremental pressures were then applied with the axial jet in order to determine the linear trend of pressure per volt for each transducer. This process was repeated with the tygon tubing attached to the negative sides of the transducers to ensure that negative voltages still fall along the positive linear trend. This is important since some pressures sensed by the five-hole probe may less than the reference pressure, which is the ambient pressure. The resulting calibrations for the transducers are shown in Table 3-1.
Table 3-1: Five-Hole Probe Pressure Transducer Calibration

<table>
<thead>
<tr>
<th>Transducer #</th>
<th>Diaphragm #</th>
<th>Max Pressure [kPa]</th>
<th>Gain [mV / V]</th>
<th>Pot</th>
<th>Calibration [Pa / Volt]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12</td>
<td>0.14</td>
<td>50</td>
<td>9.74</td>
<td>17.039</td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>0.087</td>
<td>50</td>
<td>7.82</td>
<td>17.085</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>0.087</td>
<td>50</td>
<td>11.93</td>
<td>17.043</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>0.087</td>
<td>50</td>
<td>10.14</td>
<td>17.094</td>
</tr>
<tr>
<td>5</td>
<td>10</td>
<td>0.087</td>
<td>50</td>
<td>8.81</td>
<td>17.061</td>
</tr>
</tbody>
</table>

A “non-nulling” method was selected for use with this probe during the research. This requires that before the probe can be used in an unknown flow, it first must be calibrated within a known flow to create a database of non-dimensional coefficients that can be used to describe flow angles, total pressure, and static pressure. The known flow is created by the Hammond axial jet. The five-hole pitot probe is mounted in an aluminum frame containing two rotational traverses, Figure 3-5. These traverses will rotate the probe through the yaw plane and pitch plane pictured in Figure 3-2. The aluminum frame is c-clamped to the equipment rack containing the five-hole probe transducers and amplifiers. This sets the probe at an easily accessible height. The axial jet is mounted on a hydraulic scissor jack, which allows the jet to be raised or lowered in order to center the probe vertically in the flow. Once the probe is located at 0° in its pitch plane, 0° in its yaw plane, and centered vertically, it must then be situated horizontally in the flow. Horizontally,
the five-hole probe is offset slightly to one side. This allows access for a pitot-static probe to also be inserted into the flow. The pitot-static probe will provide dynamic pressure of the flow exiting the jet so a velocity can be derived. Figure 3-6 shows the plumbing of the five-hole pitot probe and the pitot-static probe used for calibration.

Figure 3-5: Rotational Traverse Set-Up for Calibration

Figure 3-6: Five-Hole Pitot Probe Calibration Set-up
LabView was used to control the two rotational traverses, record the time-averaged voltages of the five-hole pitot probe and the pitot-static probe and write it to a text file. Figure 3-7 displays a screenshot of the LabView program “5-Hole Probe Calibration.vi”, designed by Brian Wallace\(^\text{14}\). The two main tabs are the DAQ Control tab and the Motor Control tab shown in Figure 3-8. Several values need to be changed under the DAQ tab before calibration can begin. The “Scan Rate (Hz)” was reduced to 500 Hz, while the “Max # of scans to write to file” was increased to 5000. This results in a sampling time of ten seconds at a rate of 500 Hz. Also, the “Wait Time between write and motor movements (ms)” was increased to 60000 ms. It was found that, due to the large tygon tube diameters and the amount of tubing, the five-hole pitot probe had a lag time of approximately 60 seconds when a drastic change in position is made with respect to flow direction. This parameter requires LabView to wait 60000 milliseconds after moving the probe before beginning to acquire data. Under the “Motor Control” tab, the only values requiring change are the “# of Motor 1 Pts” and the “# of Motor 2 Pts”. These values depend on the resolution of the calibration. In this case, the number of points used for both motors during the calibration was 7.
Figure 3-7: Screenshot of LabView Program used for Five-Hole Pitot Probe Calibration

Figure 3-8: Enlarged DAQ Control Tab and Motor Control Tab of LabView Program
Once a file is named to save the time-averaged voltages to, LabView initiates by moving the probe to -30° yaw and -30° pitch through the rotational traverses. After waiting 60 seconds, it acquires the transducer voltages for 10 seconds at 500 Hz and writes it to a file. The program has been designed to complete a pitch sweep in 10° increments before moving 10° to the next yaw location. This results in 49 calibration points from which four non-dimensional calibration coefficients are extracted.

A Matlab code created by Remaly serves to process the calibration data and calculate the non-dimensional coefficients utilizing Equations 3.1 to 3.4. The $P$ with a subscript number refers to the pressure sensed by the specific port on the probe. The numbers reference the specific port and were defined previously in Figure 3-2. $C_{p_y}$ refers to the flow angle within the yaw plane with respect to the probe tip. $C_{p_p}$ refers to the flow angle with the pitch plane with respect to the probe tip. The coefficients $C_{p_{static}}$ and $C_{p_{total}}$ are used to determine the velocity magnitude. $P_s$ and $P_T$ correspond to the actual static pressure and total pressure of the calibration flow, which would be acquired from the pitot-static probe. The code and the resulting calibration file can be found in Appendix A.

\[
C_{p_y} = \frac{P_2 - P_3}{P_1 - \frac{1}{4}(P_2 + P_3 + P_4 + P_5)} \quad \text{Equation 3.1}
\]

\[
C_{p_p} = \frac{P_4 - P_5}{P_1 - \frac{1}{4}(P_2 + P_3 + P_4 + P_5)} \quad \text{Equation 3.2}
\]

\[
C_{p_{static}} = \frac{\frac{1}{4}(P_2 + P_3 + P_4 + P_5) - P_s}{P_1 - \frac{1}{4}(P_2 + P_3 + P_4 + P_5)} \quad \text{Equation 3.3}
\]

\[
C_{p_{total}} = \frac{P_1 - P_T}{P_1 - \frac{1}{4}(P_2 + P_3 + P_4 + P_5)} \quad \text{Equation 3.4}
\]

Figure 3-9 displays the yaw versus pitch coefficient for the probe. Ideally, this would be symmetric about the x and y axes. However, due to small manufacturing errors in the pressure port locations, it is not. During set-up for calibration, every effort was made to zero the probe in
its pitch and yaw plane with the axial jet. None of the $0^\circ$ cases line up perfectly with their respective axis, which could be due to small errors in zeroing the placement of the probe. This needs to be taken into account when discussing accuracy of the probe.

Figure 3-10 and Figure 3-11 refer to the change in total pressure and static pressure sensed as the probe is rotated. Each yaw angle is represented by the different line series. The $\pm 30^\circ$ yaw cases are the minimum group on the graphs. The $\pm 20^\circ$ yaw cases are located in the middle. There is very little difference between the $\pm 10^\circ$ yaw and the $0^\circ$ yaw cases. They are the maximum group. The pitch angle is represented on the x-axis. Notice that at small angles the total pressure coefficient is very small. This relates to port ‘1’ of the probe capturing the entire total pressure.
3.2.3 Post-Processing

When the five-hole pitot probe was used in an unknown flow, the same LabView code was applied for data acquisition. There was only one change from the calibration format on the “DAQ Control” tab and one on the “Motor Control” tab. The “Wait Time between write and motor movements” was changed back to the default 50 ms. The initial and final positions of both motors were changed to 0 and the number of points for both motors was changed to 1. The angle
of the five-hole pitot probe was measured in reference with the axial flow of the duct using a precision digital protractor. The angle was noted in the file naming format when the file containing the time-averaged voltages was created. After acquiring the raw voltages, another Matlab code, designed by Remaly processed the data. The code and file formats can be found in Appendix B. This code reads in the raw data and the calibration data files. Using the calibration data and a double-linear interpolation scheme, flow direction and flow magnitude is determined. It then outputs several parameters of the flow including the sensed flow direction, velocity magnitude, and velocity components to a text file. The five-hole pitot probe was found to be unreliable in flow fields with low velocities or large amounts of turbulence.

3.3 **KIEL PROBE**

3.3.1 **Description**

The third instrument used to quantify flow through the duct was a United Sensors kiel probe, model KAC-18-F-12. A kiel probe consists of a total pressure port surrounded by a shroud, as shown in Figure 3-12. The diameter of this kiel probe shroud is 1.6 mm (1/16 inch) and is approximately 3.2 mm (1/8 inch) long. This makes for a probe that provides total pressure measurements with little flow obstruction. The shroud also creates a probe that is relatively insensitive to flow angles.

![Figure 3-12: Close-Up of United Sensors Kiel Probe](image)

Figure 3-12: Close-Up of United Sensors Kiel Probe
3.3.2 Calibration

The Validyne transducer (aero 615) was calibrated with the use of a precision hand-pump, slant-tube manometer, and a voltmeter. The hand-pump was connected to the positive side of the transducer as well as the manometer. The negative side was left open to atmosphere. The pressure was increased using the hand pump until the desired maximum pressure was reached on the manometer, 870.8 PA (3.5 inches of water). In order to equate this pressure to 10 volts, the gain was set at 8.28. Pressure was released and a zero was set. Incremental pressures were applied with the hand-pump up to 10 volts while recording voltage from the voltmeter. This resulted in a calibration slope of 87.77 Pascal per volt.

United Sensors reports that the KAC model kiel probe has an insensitivity range of \( \pm 52^\circ \) in the yaw plane and \(+47^\circ\) to \(-40^\circ\) in the pitch plane. The yaw and pitch plane are the same as described for the five-hole pitot probe in Figure 3-2. The kiel probe was set-up in the Hammond axial jet and the aluminum traverse frame to check the flow angle sensitivity similar to the five-hole pitot probe calibration. The plumbing set-up is displayed in Figure 3-13. The same LabView code was also used with a few changes to both the “DAQ Control” tab and the “Motor Control” tab. Under the “DAQ Control”, the “Physical Channel” pull-down menu was changed so only channel 0 and 1 was acquired. The kiel probe contains much less plumbing than the five-hole pitot probe, which relates to much less lag time. Therefore, the “Wait Time between write and motor movement” was decreased to 30000 ms. Under the “Motor Control” tab, the initial and final position of motor 1 was changed to \(-50^\circ\) and \(+50^\circ\), respectively, with 21 points. The initial and final position of motor 2 was changed to \(-60^\circ\) and \(+60^\circ\) with 25 points.
After processing the time-averaged voltages from the calibration through Matlab, the contour plot in Figure 3-14 was created to display the correlation between the actual total pressure and the total pressure sensed by the kiel probe. The blue refers to very close agreement between the two measurements. As the color becomes warmer, this represents the total pressure sensed by the kiel probe is diverging from the actual total pressure. The results are similar to the specification put out by the manufacturer. The kiel probe was found to be insensitive to flow angle in the yaw plane if it was within ±50°. The pitch plane was a little more sensitive and only allows for ±45°. It was found that positioning the kiel probe directly into the flow before starting the calibration was more difficult than with the five-hole probe because of the smaller probe size. The probe was slightly off-center for this calibration, as can be seen in Figure 3-14 by the plot not being symmetric. This will not affect the measurement accuracy as it would with the five-hole pitot probe.
Initial measurements within the forward duct of the baseline model showed the need for a local static pressure as well. The most convenient way would be to use the kiel probe for static pressure and total pressure. In order to attempt this, the kiel probe and pitot-static probe were inserted in a known flow again. The kiel probe was fixed in the pitch plane at $0^\circ$ but rotated throughout the yaw plane. After converting the voltages to pressures, pressure coefficients were calculated for each yaw angle by dividing the kiel probe sensed pressure by the dynamic pressure of the flow. Figure 3-15 displays the response of the probe to a full yaw rotation. As expected, pressures obtained within the $\pm 50^\circ$ insensitivity area had a pressure coefficient of about 1. The minimum pressure coefficient, -0.818, was found when the probe was positioned $90^\circ$ to the flow. This minimum was very different from the total pressure measurement and very sensitive, which would make it possible to relate the minimum pressure the kiel probe sensed to a local static pressure. This method will be discussed in the post-processing section of the kiel probe.
3.3.3 Post-Processing

Determining the flow velocity magnitude in an unknown flow requires two measurements to be acquired. One measurement is the maximum sensed pressure. This is the total pressure of the flow and the easiest to obtain because of the probe’s insensitivity to flow angle. The second is the minimum sensed pressure. As was found in Figure 3-15, this minimum pressure within the flow was 90° or 270° to the flow. The minimum was very sensitive to probe direction and was more difficult to determine as the flow being measured became lower velocity and/or more turbulent. The angle of the probe, when the minimum pressure is acquired, can be used to determine flow angle with some accuracy due to the probe’s high sensitivity in this position. The angle of the probe can be measured with a digital protract, as it was with the five-hole pitot probe. However, due to the very small length of the probe, this was a difficult measurement. It is suggested that if flow angle is a parameter desired to be measured by the kiel probe, a small, stiff bar be installed on the back of the kiel probe in plane with the sensing probe.
This would allow for more accurate angle measurements with the digital protractor. For even more ease measuring the angle, a protractor could be attached to the back of the probe. After angling the protractor with the reference plane, the protruding bar that was installed would display the probe angle on the protractor.

Once maximum and minimum kiel probe voltages are found and the time-averaged voltages are recorded by LabView, the data was then inserted into a spreadsheet. The voltages were converted to pressures by multiplying by the transducer calibration factor of Pascal per volt. Using the maximum and minimum pressure coefficients obtained from Figure 3-15, 1 and -0.818, as \( C_P \) in Equation 3-5 allows for the local static pressure and local velocity magnitude to be derived in Equations 3-6 and 3-7. These equations are in terms of the maximum and minimum pressure sensed by the kiel probe. The local velocity could also be determined by subtracting the calculated local static pressure from the maximum pressure sensed by the kiel probe. This will result in the local dynamic pressure of the flow.

\[
C_P (MAX/MIN) = \frac{P_{KIEL(MAX/MIN)} - P_{LOCAL}}{\frac{1}{2}\rho V^2_{LOCAL}} \tag{Equation 3-5}
\]

\[
P_{LOCAL} = \frac{P_{KIELMAX} + (1.235 \times P_{KIELMIN})}{2.235} \tag{Equation 3-6}
\]

\[
V_{LOCAL} = \sqrt{\frac{2.445(P_{LOCAL} - P_{KIELMIN})}{\rho}} \tag{Equation 3-7}
\]

3.4 Hammond Facilities

3.4.1 Hammond Wind Tunnel

A portion of the forward flight and hover measurements were acquired in the Penn State Hammond Building wind tunnel located in rooms 6 and 8. A side view schematic of this tunnel can be seen in Figure 3-16. It is a closed circuit, single return, low speed, atmospheric facility.
Flow is powered by a 1.22 meter (4 feet) diameter fan belt-driven by a 25 hp motor suspended below the fan casing. After flow passes through the contraction section with an area ratio of 9.9:1, it can reach a maximum velocity of 44 m/s (145 ft/s) within the test section. This long rectangular test section has dimensions of 0.61 meters (2 feet) wide by 0.91 meters (3 feet) tall by 6.1 meters (20 feet) long. The corners are filleted. The turbulence management consists of 5 screens, 1 perforated plate, and a 0.15 meter (6 inch) long honeycomb section. The turbulence intensity has been measured to be 5% at 20 ft/s to 0.3% at 140 ft/s. For this particular research, all experiments were performed in the upstream portion of the test section.

![Figure 3-16: Side View of Hammond Wind Tunnel](image)

3.4.2 Hammond Force and Moment Balance

The Hammond force and moment balance, Figure 3-17, is a five-channel, three-component platform balance that is suspended above the forward portion of the Hammond wind tunnel test section. It is capable of measuring lift, drag, and pitching moment through five load cells. Two load cells are placed vertically in plane to measure lift. They are known as the near and far lift channels with “near” being the left side if looking in the direction of the flow. In the same plane as the lift load cells, positioned horizontally, are two drag load cells also referred to as near and far drag. The fifth load cell is positioned vertically on the back of the balance to
measure pitching moment. A model is attached to the balance through one or two suspended struts. The double strut configuration was used with the model in this thesis.

![Figure 3-17: Hammond Force and Moment Plate Balance](image)

Located just downstream of the pitching moment load cell is a stepper motor mounted on the balance. A threaded rod with a symmetric airfoil cover and a pivot point was inserted into the stepper motor and attached to the rear of the model. This allows for an angle of attack adjustment between +20° and -20°. An external controller is used to power the stepper motor.

Due to the location of the multiple load cells, cross-talk occurs between all the channels when in loading. The load paths do not pass through a focal point. Therefore as an example, even if a pure lift load is being applied, the drag and pitching moment load cells will also sense loading. Calibration was used to eliminate the cross-talk. With the model attached at an angle of attack of 0°, incremental loads in pure lift, pure drag, and pure pitching moment in order to calibrate the balance. Pure lift was applied by suspending weight from the center-body of the model. Pure drag was applied by centering a pulley downstream of the model. String was attached to the rear of the model, run over the pulley, and attached to weight. Special care must be taken to ensure the string that travels from the model to the pulley has no offset downstream or laterally. Otherwise, forces other than drag will be applied. Pure pitching moment is applied by
hanging weight from the aft fan and placing a pulley above the forward fan to lift. Again, the
pulley must be placed so that there is no offset in the string from the duct to the pulley
downstream or laterally. This would complete the calibration of the balance. As described in
Section 5.2.4 of the thesis of Leighton Myers, two 3x3 influence coefficient matrixes, seen in
Table 3-2, were created using the calibration results for the near and far load cells.\(^{(11)}\) Equation
3-8 describes the matrix in more detail. It would be expected that the main diagonal of these
matrices would be the most dominate due to these relating to the pure lift, drag, and pitching
moment forces. However, it is seen that pitching moment has a large effect on the drag
measurement. The large amount of cross-talk is seen in the high coefficient at \(A_{PD}\). After creating
the ICMs, the same techniques as with the calibration are used to create mixed loads in lift, drag,
and pitching moment in order to test the coefficient matrix.

<table>
<thead>
<tr>
<th>Table 3-2: Hammond Balance Influence Coefficient Matrixes</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Near Side Influence Coefficient Matrix</strong></td>
</tr>
<tr>
<td>( ICM_{NEAR} = \begin{bmatrix} -0.22334 &amp; 0.13549 &amp; -0.00642 \ -0.00767 &amp; -0.25320 &amp; -0.00078 \ -0.00499 &amp; -0.25644 &amp; 0.01501 \end{bmatrix} )</td>
</tr>
</tbody>
</table>

| **Far Side Influence Coefficient Matrix**            |
| \( ICM_{FAR} = \begin{bmatrix} -0.22169 & 0.12400 & -0.00597 \\ -0.02726 & -0.24309 & -0.00147 \\ -0.00500 & -0.25644 & 0.01051 \end{bmatrix} \) |

\[ ICM = \begin{bmatrix} A_{LL} & A_{LD} & A_{LP} \\ A_{DL} & A_{DD} & A_{DP} \\ A_{PL} & A_{PD} & A_{PP} \end{bmatrix} \]  

Equation 3-8
3.5 ACADEMIC PROJECTS BUILDING FACILITIES

3.5.1 Academic Projects Building Wind Tunnel

Most of the forward flight and hover measurements were acquired in the Penn State Academic Projects Building, APB, wind tunnel located in room 117. A top view schematic of this tunnel can be seen in Figure 3-18. It also is a closed circuit, single return, low speed, atmospheric facility. The overall dimensions of the tunnel are 22.9 meters (75 feet) long by 7.6 meters (25 feet) wide. Flow is powered by a 1.8 meter (6 feet) diameter fan shaft-driven by a 300 hp motor located external to the second corner of the tunnel. After flow passes through the contraction section with an area ratio of 9.6:1, it can reach a maximum velocity of 67 m/s (220 ft/s) within the test section. The rectangular test section has dimensions of 1.5 meters (5 feet) wide by 1.0 meters (3.25 feet) tall by 1.9 meters (6.1 feet) long. The corners are filleted. The turbulence management consists of 1 dust screen to reduce recirculating particulates, 1 perforated plate to eliminate large scale air flow separation, a 0.15 meter (6 inch) long honeycomb section to reduce swirl, and 4 screens varying in mesh size to remove shear layers in the air flow. The turbulence intensity has been measured to be 0.045% at 150 ft/s. More information on the APB wind tunnel is contained in the theses of Swan and Germanowski.

Figure 3-18: Top View of the Academic Projects Building (APB) Wind Tunnel
3.5.2 Academic Projects Building Force and Moment Balance

The APB force and moment balance, Figure 3-19, is a six-channel, six-component pyramid balance that was placed under the APB wind tunnel test section for forward flight experiments, as well as outside of the tunnel for out-of-ground-effect (OGE) hover experiments. It is capable of measuring lift, drag, side force, pitching moment, rolling moment, and yawing moment through six load cells. Two struts extend from the table and are used for attachment of models. They are capable of a maximum separation distance of 0.91 meters (3 feet). Due to the pyramid type, if the model is mounted in the top holes of the struts, all forces and moments will act through their respective load paths and crosstalk between channels will be minimized. However, the dual-ducted fan model was unable to be attached at the top of the struts due to the size requirement of the bolt used for attachment. The model had to be mounted through larger holes in the struts that are located lower than the focal point. This distorts the load paths and creates crosstalk, as will be seen in the calibration data presented later. The APB balance also supports the use of a motorized angle of attack bar, side slip adjustment, and height adjustment.

![Figure 3-19: APB Force and Moment Pyramid Balance](image-url)
As with the Hammond force and moment balance, calibration with known forces and moments is required to account for crosstalk and determine the correct conversion from load cell voltages to the corresponding forces and moments. Calibration was performed outside of the APB wind tunnel. Six foot tall struts were bolted to the base of the balance at each corner. This would provide locations for pulleys to be mounted. The model was attached to the balance struts upside down. In this configuration, the model downwash would be directed toward the ceiling of the tunnel and not towards the balance table. With the model attached to the balance struts and at an angle of attack of 0°, pure forces and moments were applied using weights, string, and pulleys. Once the pure forces and moments were applied and an influence coefficient matrix was produced, several cases of mixed forces and moments were applied to check for error.

A pure lift load was applied by simply looping a section of string around the center-body of the model. The string was positioned along the center-line. A weight hanger was hung from the string directly in-line with the balance struts. Incremental weights were applied to the hanger and then removed in the same way they were added. LabView was used to acquire the voltages of all the load cells at each weight increment. After several pure lift calibrations, it was learned that the balance reaction to small lift loads was much improved if seven pounds of weight was applied directly to the table. These weights were carefully placed in the center of the table and remained throughout the experiments in and out of the wind tunnel.

Before a pure drag load could be applied, a pulley must be installed on the rear of the balance. To do this, angle iron was c-clamped to the struts at the rear of the balance that were bolted to the base. String was attached to the aft-most portion of the model and run over the pulley. The position of the angle iron and pulley were moved until the position of string remained unchanged vertically or horizontally from the model to the pulley. At weight hanger was then attached to the string after the pulley and incremental weights were applied and removed.
A pulley to the side of the balance was required for the pure side force calibration. The pulley would be placed directly in-line with the balance struts. The string was attached to the balance strut at the point where the model attached. The pulley was positioned so that the string’s position didn’t change vertically or horizontally from the attachment to the balance strut to the pulley. Incremental weights were then applied and removed.

A pure pitching moment was applied by looping a string over the forward-most part of the model. The string was positioned along the center-line of the model and a weight hanger was placed on the string. In order to create a pure pitching moment, the aft part of the model must be lifted. This was done by attached angle iron down the sides of the balance on the base-mounted struts. Another piece of angle iron was then c-clamp to them and ran directly across the aft-most part of the model. A pulley was mounted to this so that a string would run from the aft-most part of the model up to the pulley without changing position. The moment arm, distance between the two strings, must be measured in order to calculate the known moment applied. Due to the inverted position of the model, this created a pure nose-up pitching moment.

A pure rolling moment was applied by placing a loop of string on the right side of the model as close as possible to the balance strut. On the left side, angle iron and a pulley was set-up so a string could be place as close as possible to the left balance strut and pull up. Again, it was ensured that the position of the string didn’t change while traveling from its attachment on the model to the pulley. The moment arm must be measured between the two strings so that the known moment can be calculated.

Finally, a pure yaw moment was applied using two pulleys. Using the angle iron down the sides of the balance, one pulley was placed to the left of the forward duct and one to the right of the aft duct. The pulley and string were aligned so that the force would be applied in-line with forward and aft motors, or center of the two ducts. Measuring the distance between the center points of the two ducts would provide a moment arm for the yawing moment.
After all the pure loadings had been performed, an influence coefficient matrix was calculated. Equation 3-9 illustrates the 6x6 matrix required for the APB balance. The term at (1,1) represents the affect of drag on the drag channel and term (2,1) represents the affect of side force on the drag channel and so on. The remaining terms are ‘L’ for lift, ‘R’ for rolling moment, ‘P’ for pitching moment, and ‘Y’ for yawing moment. For a pyramid balance, the main diagonal is expected to be dominate while the other terms remain infinitesimal. Table 3-3 displays the ICM obtained for the experiments presented in this thesis. For the most part, the main diagonal is dominate. However, due to the mounting location of the model, there is additional cross-talk. This is mostly seen in the affect the pitching moment has on the drag channel.

\[
\begin{bmatrix}
A_{DD} & A_{DS} & A_{DL} & A_{DR} & A_{DP} & A_{DY} \\
A_{SD} & A_{SS} & A_{SL} & A_{SR} & A_{SP} & A_{SY} \\
A_{LD} & A_{LS} & A_{LL} & A_{LR} & A_{LP} & A_{LY} \\
A_{RD} & A_{RS} & A_{RL} & A_{RR} & A_{RP} & A_{RY} \\
A_{PD} & A_{PS} & A_{PL} & A_{PR} & A_{PP} & A_{PY} \\
A_{YD} & A_{YS} & A_{YL} & A_{YR} & A_{YP} & A_{YY}
\end{bmatrix}
\]

Equation 3-9

| ICM = | -1.0774 | -0.0045 | -0.0202 | 0.0026 | -0.0181 | 0.0002 |
| -0.0102 | 0.9595 | 0.0137 | 0.0015 | -0.0044 | -0.0043 |
| -0.0277 | 0.0126 | 2.4772 | 0.0021 | -0.0531 | 0.0192 |
| -0.4096 | 1.0369 | -0.0723 | -3.8712 | 0.1860 | 0.1128 |
| 2.1068 | 0.1566 | -1.3383 | 0.0555 | 13.3791 | -0.2010 |
| 0.1253 | 0.3292 | -0.1534 | 0.0616 | 0.1702 | -4.0725 |

Table 3-3: APB Force and Moment Balance Influence Coefficient Matrix
Chapter 4

BASELINE MODEL RESULTS AND DISCUSSION

Extensive measurements were completed on the baseline dual-ducted fan model in order to better understand the performance of a full-sized vehicle in hover and forward flight. Flow measurements were obtained with the mini-vane anemometer, five-hole pitot probe, and kiel probe discussed in Chapter 3. Loads and moments were measured in hover and forward flight using the Penn State wind tunnels also discussed in Chapter 3 and their respective force and moment balances. Flow visualization was performed in the wind tunnels using a single smoke wire in order to visualize the flow characteristics into the forward fan of the model while in forward flight. As will be shown in this chapter, these measurements helped create a database for this unique vehicle and introduced more questions relative on improving upon its performance.

4.1 HOVER

4.1.1 Flow Measurements

The first measurements on the dual-ducted fan model were axial flow velocity, flow vertically through the duct, while in a static hover operating out of ground effect. The model was mounted on the APB balance struts, which were attached to a 2x4 lumber base. The model was mounted upside so that outflow from the fans was directed towards the ceiling. This would ensure the fans were operating completely out of ground effect. It was also set at an angle of attack of 0°. The mini-vane anemometer was placed above and below the forward and aft fan at several locations with the axis of the vane parallel with the duct axis. Flow velocities will be referred to as axial velocities; however, due to the shroud around the anemometer, some off-axis flow is also captured. Figure 4-1 illustrates the locations measurements were obtained with the
Using the time-averaging capabilities of the anemometer, velocity magnitudes were collected for ten seconds at each location and for different rotor RPMs.

Presented in Figure 4-2 are the time-averaged inflow velocities obtained by the mini-vane anemometer above the fans while the model was hovering at 6000 RPM. Figure 4-2(a) displays the aft fan, while Figure 4-2(b) is the forward fan. Each graph contains velocity magnitudes when a single fan is running and when both are running. The locations for these measurements are along the center-line, or the longitudinal axis of symmetry, of the model.

Velocities above the forward fan were measured to be greater than those above the aft fan for the entire area. This would suggest the forward fan will produce more thrust than the aft fan and, therefore, will produce a nose-up pitching moment. Most likely, the reason for the increased velocities above the forward fan is due to the geometry of the duct or location of the rotor within the duct. As discussed in Chapter 2, the geometry of both rotors is nearly identical. The velocities also vary span-wise. The front-most part of the forward fan was found to experience
the greatest inflow velocities. The inflow velocities then decreased as the rear-most part of the forward fan is approached. This can most likely be attributed to the lack of motor support struts in the front half of the forward fan, refer to Figure 4-1. These struts introduce a blockage to the flow, and their absence in the front half of the forward fan allows the flow easier passage.

There is little difference in velocity when comparing a single fan running to both fans running except at the aft-most portion of the forward fan. It seems that the aft fan may rob the forward fan of some flow while in hover. The velocities decrease in this portion of the forward fan while velocities in the forward-most section of the aft fan are seen to increase slightly.

![Figure 4-2: Axial Velocity Above the Rotor along the Baseline Model Centerline Measured by the Mini-Vane Anemometer at 6000 RPM in Hover: (a) Aft Fan, (b) Forward Fan](image)

Figure 4-3 is similar to the previous figure except it presents the velocity magnitudes measured below the two fans. When comparing these graphs, it should be noted that velocity was very sensitive to location when measure near the edges of the ducts (±1.00). There may be error at these locations due to small changes in placement of the anemometer.

Although higher inflow velocities were seen above the forward fan in Figure 4-2, Figure 4-3 now illustrates that the outflow velocities, or velocities below the rotor, are slightly higher in the aft fan than the forward fan. The reduced outflow velocities in the forward fan, most
prevalent when only the forward fan is running, may be contributed to swirl in the fan. Referring again to Figure 4-1, it can be seen that there are no support struts in the front half of the forward fan. The argument can be made that these struts also act as stator vanes to straighten the flow. Their absence in the front half of the forward fan increases the amount of swirl in the forward fan and decreases the axial flow velocity exiting the duct.

Flow velocities were found to change drastically outside of the duct, especially around the duct edges. It was decided that holes would be drilled in one side of the forward fan of the model approximately 13 mm (0.5 in.) above and below the rotor. This would allow for a five-hole pitot probe to penetrate the duct and measure velocities within the duct. The probe would then be closer to the rotor and would provide a more accurate velocity measurement of the flow encountered by the rotor. Use of the five-hole pitot probe would also allow for a flow angle to be determined.

Figure 4-4 explains the locations measurements were obtained within the front duct with the five-hole pitot probe. Each point is lettered and can be referenced to Table 4-1 for a detailed location within the duct. The points enclosed with parenthesis represent locations below the
rotor. Also note that the forward rotor rotates in the counter-clockwise direction. Most of the measurements occur around 70% of the rotor radius. The reasoning behind this was because it was assumed that the radial location of maximum thrust was approximately located at 75% of the rotor radius. By obtaining points in this region around the rotor disk, azimuthal comparisons could be made where the rotor is expected to be at its peak performance. One point is located at the forward 30% radius providing insight of changes in the flow closer to the hub.

Figure 4-4: Five-Hole Probe and Kiel Probe Locations for Flow Measurement within the Forward Duct
Before presenting the five-hole pitot probe hover measurements, it is best to define the coordinates that will be used. Figure 4-5 displays the model coordinate system. The model is positioned at an angle of attack in the figure; however, in hover the model was at an angle of attack of 0°. The probe was positioned with respect to the body-fixed coordinate system such that the sensing part of the probe was parallel with the Z_b axis, or the axis of the duct. Thus the flow angle \( \theta \) was defined with respect to the body fixed coordinate system. The flow angle \( \theta \) is defined in the \( X_b-Z_b \) plane, positive toward the 180° azimuth, from the duct axis.

<table>
<thead>
<tr>
<th>Point Label</th>
<th>Type</th>
<th>Longitudinal Distance (in)</th>
<th>Lateral Distance (in)</th>
<th>% Radius</th>
<th>Azimuthal Angle (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Inflow</td>
<td>3.25</td>
<td>1.75</td>
<td>78.75</td>
<td>151.7</td>
</tr>
<tr>
<td>B</td>
<td>Inflow</td>
<td>3.25</td>
<td>0</td>
<td>69.33</td>
<td>180</td>
</tr>
<tr>
<td>C</td>
<td>Inflow</td>
<td>3.25</td>
<td>1.75</td>
<td>78.75</td>
<td>208.3</td>
</tr>
<tr>
<td>D</td>
<td>Inflow</td>
<td>1.5</td>
<td>2.875</td>
<td>69.18</td>
<td>117.55</td>
</tr>
<tr>
<td>E</td>
<td>Inflow</td>
<td>1.5</td>
<td>0</td>
<td>32</td>
<td>180</td>
</tr>
<tr>
<td>F</td>
<td>Inflow</td>
<td>1.5</td>
<td>2.875</td>
<td>69.18</td>
<td>242.45</td>
</tr>
<tr>
<td>G</td>
<td>Inflow</td>
<td>2.9375</td>
<td>1.5</td>
<td>70.36</td>
<td>27.05</td>
</tr>
<tr>
<td>H</td>
<td>Inflow</td>
<td>2.9375</td>
<td>0</td>
<td>62.67</td>
<td>0</td>
</tr>
<tr>
<td>I</td>
<td>Inflow</td>
<td>2.9375</td>
<td>1.5</td>
<td>70.36</td>
<td>332.95</td>
</tr>
<tr>
<td>J</td>
<td>Outflow</td>
<td>3.25</td>
<td>1.75</td>
<td>78.75</td>
<td>151.7</td>
</tr>
<tr>
<td>K</td>
<td>Outflow</td>
<td>3.25</td>
<td>0</td>
<td>69.33</td>
<td>180</td>
</tr>
<tr>
<td>L</td>
<td>Outflow</td>
<td>3.25</td>
<td>1.75</td>
<td>78.75</td>
<td>208.3</td>
</tr>
<tr>
<td>M</td>
<td>Outflow</td>
<td>1.5</td>
<td>2.875</td>
<td>69.18</td>
<td>117.55</td>
</tr>
<tr>
<td>N</td>
<td>Outflow</td>
<td>1.5</td>
<td>0</td>
<td>32</td>
<td>180</td>
</tr>
<tr>
<td>O</td>
<td>Outflow</td>
<td>1.5</td>
<td>2.875</td>
<td>69.18</td>
<td>242.45</td>
</tr>
<tr>
<td>P</td>
<td>Outflow</td>
<td>2.75</td>
<td>1.75</td>
<td>69.54</td>
<td>32.47</td>
</tr>
<tr>
<td>Q</td>
<td>Outflow</td>
<td>2.75</td>
<td>0</td>
<td>58.67</td>
<td>0</td>
</tr>
<tr>
<td>R</td>
<td>Outflow</td>
<td>2.75</td>
<td>1.75</td>
<td>69.54</td>
<td>327.53</td>
</tr>
</tbody>
</table>
Figure 4-6 presents the velocity magnitude and the three velocity components above the rotor as measured by the five-hole pitot probe while the dual-ducted fan model was in hover at 4000 RPM and 6000 RPM. U_B is the velocity component in the direction of X_b, V_B is the velocity in the direction of Y_b, and W_B is the velocity in the Z_b direction. The measurement locations are all above the forward rotor and approximately at 75% of the rotor radius. The points coincide with point A to D and F to I from Table 4-1. The azimuthal angle of 180° refers to the forward-most section of the fan.

In Figure 4-6, the velocity magnitude and axial velocity is nearly the same for both rotor RPMs. Where these two differ most is in the forward half of the fan. Although these velocities are measured above the rotor, the lack of struts in the front of the fan may be slightly affecting the
flow above the rotor. The other two velocities components are small compared to the axial velocity. This is beneficial because they would represent a loss of useful work.

![Velocity Magnitude and Components](image)

**Figure 4-6: Velocity Magnitude and Components Measured by the Five-Hole Pitot Probe Above the Forward Rotor of the Baseline Model in Hover: (a) 4000 RPM and (b) 6000 RPM**

The $U_B$ component, or downstream velocity, is positive in the front half of the fan (90° to 270°) and negative in the aft half (0° to 90° and 270° to 360°). This is as expected and would correspond to the slipstream converging into the duct. For an ideal isolated rotor, $U_B$ would be expected to be symmetric; $U_B$ at 0° would be equal to but the negative of 180° and would measure to be 0 at 90° and 270°. After placing the duct around the rotor, the slipstream doesn’t converge as sharply in the rear section of the front fan due to duct geometry. The rotor may also be inducing swirl in the flow above, which would cause the $U_B$ velocities at 90° and 270° to be non-zero and of opposite sign.

The $V_B$ component of an ideal isolated rotor should consist of a velocity maximum at 90° and a velocity minimum at 270°. This cross-stream velocity would be zero at 0° and 180°. This would also be consistent with the slipstream converging into the duct. There may be some
small changes due to rotation induced by the rotor. This pattern is nearly seen in the measurements obtained in the forward duct above the rotor. However, the points located approximately at 118° and 242° behave opposite of what is expected for an isolated rotor. At this time, this cannot be explained. These two locations should have repeat measurements performed to if this trend is repeatable.

Figure 4-7 shows the velocity components below the forward rotor while in hover approximately at 75% of the rotor radius. The velocity magnitude and axial velocity, W_B, are very similar to their counterparts in the inflow. The agreement between the magnitude and W_B is better in the outflow than the inflow. This is unexpected. It was hypothesized that the lack of struts in the front portion of the forward duct would cause the off-axis velocity components, U_B and V_B, to be more prevalent in the outflow measurements due to swirl induced in the flow by the rotor. This would result in a reduction of axial velocity and create a greater difference between the axial velocity and the magnitude.

![Figure 4-7: Velocity Magnitude and Components Measured by the Five-Hole Pitot Probe Below the Forward Rotor of the Baseline Model in Hover (a) 4000 RPM and (b) 6000 RPM](image)

Although the off-axis velocities, U_B and V_B, were not as large as was expected after examining the inflow, they do describe swirl induced in the flow due to the rotor. Both
components display a counter-clockwise rotation that can be linked to the rotor rotation. \( U_B \) has maximums at 90° and 270°, while nearly zero at 0° and 180°. The cross-stream component, \( V_B \), has a maximum at 180° and a minimum at 0°. They are both nearly zero at the rear portion of the forward fan, which still suggests the struts affect the flow characteristics.

The velocity components help to describe the slipstream through the duct, but the angle \( \theta \), in the model’s plane of symmetry is described in Figure 4-5 and demonstrates this much better. The angles measured by the five-hole pitot probe are displayed for the entire azimuth at 75% of the rotor radius above and below the rotor in Figure 4-8 for the two RPMs examined. The slipstream is seen to converge very quickly in the front portion of the fan at 25°, while the convergence is much more subtle in the rear portion at approximately 6°. The angles along the center-line only are illustrated in Figure 4-9. The inflow shows the slipstream convergence into the duct. The front 70% displays the sharp angle which lessens as the center of the duct is approach, as seen by the angle at the front 30%. The rear 70% suggests that this part of the slipstream has nearly finished convergence. Below the rotor, the front and rear 70% points demonstrate that the outflow is close to being purely axial. At the front 30% in the outflow, some divergence is seen, which can most likely be related to interference caused by the hub and motor.

![Figure 4-8: Downstream Flow Angle, \( \theta \), Measured by the Five-Hole Pitot Probe within the Forward Fan of the Baseline Model Operating in Hover as a function of Azimuthal Angle: (a) Above the Rotor, (b) Below the Rotor](image-url)
4.1.2 Force and Moment Measurements

The dual-ducted fan model was integrated with APB force and moment balance outside of the wind tunnel. The model was mounted upside-down so that the fans would blow toward the ceiling eliminating any ground effect. The balance could then be used to measure the total thrust and pitching moment of the model while in a steady, level hover.

Unfortunately, the set-up does not allow rotor thrust to be separated from duct thrust. The total thrust reported in Figure 4-10 is the sum of the thrust produced by both rotors and the thrust produced by the two ducts. Increasing the rotor RPM increases the thrust nearly proportional to the angular rotor speed squared, $\Omega^2$. The maximum achievable RPM with the described motor and electronics is 10,000, which corresponds to the tip mach number of 0.36. At this speed, a maximum thrust of 35.6 N is produced.
Figure 4-10: Total Thrust (the sum of Rotor Thrust and Duct Lift) for Various Rotor RPMs Measured by the APB Balance while operating in Hover

The thrust was non-dimensionalized using the US customary definition of thrust, Equation 4-1, as presented by Leishman.\(^{(4)}\) The total thrust, \(T\), is non-dimensionalized by the air density, \(\rho\), the sum of the disk area of both fans, \(A\), and the rotor tip speed, \(V_{tip}\). Figure 4-11 presents the thrust coefficients across the rotor RPM range. The thrust coefficient of an isolated rotor would be constant across the RPM range. Since the rotor thrust and duct thrust is summed here, it is assumed the ducted rotor doesn’t follow the isolated rotor trend due to the duct thrust increasing as rotor RPM increases, which can be related to higher outflow velocities.

\[
C_T = \frac{T}{\rho A V_{tip}^2}
\]  
Equation 4-1
The pitching moment of the dual-ducted fan in hover is displayed in Figure 4-12. A positive pitching moment refers to a nose-up moment. The flow velocity measurements made by the mini-vane anemometer for both fans suggested the two fans were very similar. The pitching moment measured by the APB balance reinforces these measurements. It is nearly zero except for at 10,000 RPM. A slight nose-down pitching moment is experienced. This confirms that, at least in hover, both fans and ducts produce nearly the same thrust.
4.2 **FORWARD FLIGHT**

Axial flow through a ducted fan has been often studied and is reasonably well understood. Placing a duct in edgewise flight is much more complicated and is yet to be completely understood. The dual-ducted fan model this thesis is addressing is new to the industry and has been examined in less detail. It was felt the best approach to such a new problem was to first obtain qualitative data about the flow around the model while in forward flight.

4.2.1 **Flow Visualization**

Qualitative data about the flow around the model in forward flight was obtained via flow visualization techniques. A single, vertical plane of streamlines was created by a smoke wire. A thin diameter wire was placed in-plane with the model’s center-line, or axis of symmetry. Mineral oil was then placed on the wire with the use of a cotton swab. In order to evaporate the oil into smoke streamlines, current was passed through the wire by an external power supply. The current of the power supply was varied depending on the speed of the wind tunnel flow. If the power supply is too low, the smoke lines last longer but will be very faint. If the power supply is too high, the smoke lines will be more defined but the oil will evaporate very quickly and the wire is in risk of breaking. It was found that operating at the lowest rotor speeds and wind tunnel speeds resulted in the best images of the streamlines.

Figure 4-13 displays smoke wire flow visualization for three different configurations of the baseline dual-ducted fan model. All were performed at a wind tunnel velocity of 2 m/sec (6.5 ft/sec) from right to left on the image. The top left configuration consists of the model being set at an angle of attack of -3° and the rotors are off. Flow appears to be attached on the upper surface of the model fuselage. With the rotors off, it appears there is no flow entering the ducts. On the lower surface, flow becomes separated immediately. This is not an issue since when the rotors are turned on and flow exits the duct; it will change flow on the lower surface completely.
Figure 4-13: Streamlines from a Single Vertical Smoke Wire Placed at the Model Centerline

The configuration of the top right is still at an angle of attack of -3°, but the rotors have been turned on at 2500 RPM resulting in an advance ratio of 0.065. Flow was seen being turned and entering into the ducts. The downwash below the fans distorts most of the streamlines on the lower surface. It is very interesting to observe streamlines that enter the front portion of the forward duct from below the fan. The pressure below the fan is lower than the pressure on the lower surface of the fuselage suggesting the fan is ineffective at producing thrust in this area. The cause for this would be separation of incoming flow over the leading edge radius of the duct.

The bottom flow visualization image was taken when the model was angled at +3° and the rotors were operating at 2500 RPM. The only noticeable change in this image from the last one is the streamlines on the lower surface. The streamlines are undisturbed under most of the forward fan. The overall downwash also appears much weaker than when the model was positioned at an angle of attack of -3°.
The angle of attack of the model was decreased to -10°. This was done in an attempt to reduce the amount of separation over the leading edge radius of the forward duct. Figure 4-14 and Figure 4-15 display close-ups of the area in question at two advance ratios at this angle of attack. In Figure 4-14, the rotor RPM is set at 2000, while the wind tunnel flow is 2 m/sec (6.5 ft/sec) from right to left. This relates to an advance ratio of 0.08. Streamlines show that the flow is separating over the front leading edge radius of the forward fan. The separation bubble extends nearly half of the rotor radius towards the hub. This renders that portion of the fan ineffective. In Figure 4-15, the rotor RPM is set at 2000 RPM and the wind tunnel velocity is 6.1 m/sec (20 ft/sec) from right to left, which is an advance ratio of 0.24. The separation bubble off the leading edge radius of the duct has now grown to the point that it encompasses over half the rotor disk. The streamlines don’t come in contact with the rotor until downstream of the rotor hub.

Figure 4-14: Close-up of Flow Separation over the Forward Duct Lip along the Centerline at an Angle of Attack of -10° and Advance Ratio of 0.08

Figure 4-15: Close-up of Flow Separation over the Forward Duct Lip along the Centerline at an Angle of Attack of -10° and Advance Ratio of 0.24
4.2.2 Flow Measurements

The kiel probe and five-hole pitot probe were used to penetrate the forward duct and obtain quantitative measurements at the locations described earlier in Figure 4-4 while the model was at an angle of attack of -10°. Figure 4-16 displays the measurements obtained by the two different probes. The five-hole pitot probe, the bold, black outlined symbols, was found to be unreliable or ineffective when in low velocity flow or very turbulent flows. Therefore, data could not be collected using the five-hole probe in the area of the separation bubble off the leading edge radius. The kiel probe, the standard symbols with lines, was capable of measuring velocity magnitudes at all locations. When measurements were made with both probes, both displayed similar trends. The percent difference between the two probes ranged from as small as 3% up to 25%. One case in Figure 4-16(d) at 6000 RPM displayed a very large difference; however, this was the only point where the five-hole pitot probe was able to measure a velocity. Even though the probe was able to resolve the velocity at this one point, there may be error induced by the flow separation or other unknown factors. Typically, the percent difference between the probes was less than 20%. Some of the difference between the probe measurements could be related to the locations they measure. Although the kiel and the five-hole probe were always positioned the same azimuthally and radially, the kiel probe’s measurement point is in-plane with its shaft. The five-hole pitot probe extends 0.61 inches above the plane of its shaft when position directly into the axial flow through the duct.

The velocity magnitudes within the forward duct can be more carefully examined in Figure 4-17. The average velocity magnitude measured by both probes is shown here, when available. Otherwise, the velocity magnitude shown is only from the one probe. The standard symbols with the line are the points located at approximately 70% of the rotor radius, while the singular bolded points are at the front 30% rotor radius location. Velocity magnitude above and
below the rotor are displayed for two different wind tunnel speeds, 6.1 m/sec (20 ft/sec) and 10.7 m/sec (35 ft/sec), as well as three different rotor RPMs: 0, 4000, and 6000. This relates to advance ratios of 0.08 (6000 at 6.1 m/s), 0.12 (4000 at 6.1 m/s), 0.14 (6000 at 10.7 m/s), and 0.21 (4000 at 10.7 m/s). The model was positioned at an angle of attack of -10°.

Examining Figure 4-17 (a) and (b), velocity magnitude above the rotor, it is found that, contrary to what was observed with the flow visualization, velocity was measured at 180° on the rotor at the 30% radius and 70% radius locations. However, there is a difference of 4 m/s
between the 30% radius and 70% radius velocity at 180° in (a). This does suggest that the flow into the rotor closer to the front lip is not optimal and that there is separation. In (b), the results are similar to those in (a), but the difference between the 70% radius and 30% radius at the 180° is greater. Also, the 70% radius velocity closest to the duct lip, 151° to 208°, is lower than that measured in (a), while the 30% radius velocity is higher than in (a).

Figure 4-17: Average of Velocity Magnitudes Measured by the Kiel Probe and Five-Hole Pitot Probe within the Forward Duct of the Baseline Model: (a) Above the Rotor at $V_{WT}=6.1$ m/s, (b) Above the Rotor at $V_{WT}=10.7$ m/s, (c) Below the Rotor at $V_{WT}=6.1$ m/s, (d) Below the Rotor at $V_{WT}=10.7$ m/s.
The velocity magnitudes below the rotor are presented in (c) and (d) for the two wind tunnel velocities. Both of these plots demonstrate the effect of the separation bubble. Velocity magnitudes are much lower in the front of the fan closer to the duct lip than they are in the rear of the fan. Interestingly enough, the reverse flow seen in the flow visualization, flow being ingested into the bottom of the fan, was measured between 151° and 208°. An explanation for favorable velocity magnitudes being measured in the inflow but in the outflow could be because of the angle of the flow. If the flow is traveling through the duct at an angle, the separation bubble could be growing as flow proceeds through the duct.

Figure 4-18 presents the flow angle $\theta$, as defined in Figure 4-5, measured with the five-hole pitot probe azimuthally above, (a), and below, (b), the forward rotor. The model and tunnel conditions were set for an advance ratio of 0.08 at 6000 RPM and 6.1 m/sec (20 ft/sec). The flow angle measured in the hover case is presented by a dashed curve in each plot. The standard blue diamond symbols are located at approximately 70% the rotor radius. The single blue diamond with the bold black outline is located at 30% the rotor radius. The model was positioned at an angle of attack of -10°.

Referring to Figure 4-18 (a), the angle $\theta$ measured in forward flight follows the same trend as in hover, with the exception that in forward flight, the angle never becomes negative. For all the azimuth locations measured in forward flight, there was always a downstream sweep in flow angle. Similar to the hover case, the maximum angle in this plane was observed forward of the rotor hub and along the centerline. However, for the forward flight case, the maximum angle was measured at the 30% radial location, since the flow angle at the 70% radial centerline could not be resolved.

Figure 4-18 (b) presents the flow angle $\theta$ as a function of azimuthal angle measured below the forward rotor during the 0.08 advance ratio test condition. All of the data points shown for the forward flight case measured at the 70% radial span share a similar trend with the hover
case; however, all of the points measured were positive. As was discussed with the hover angles, these angles help to describe the swirl that is induced by the rotor in the outflow.

![Diagram showing flow angle θ within the forward fan as a function of azimuthal angle at an advance ratio of 0.08: (a) Above the Rotor, (b) Below the Rotor.](image)

Figure 4-18: Downstream Flow Angle, θ, within the Forward Fan as a function of Azimuthal Angle at an Advance Ratio of 0.08: (a) Above the Rotor, (b) Below the Rotor

The flow angle θ along the centerline of the model associated with the advance ratio 0.08 (6000 RPM, \(V_{WT}=6.1\text{m/sec}\)) is illustrated in Figure 4-19. As was discussed in Figure 4-18, the flow is always angled downstream. Passing through the rotor causes the flow to become more axial, but there is still a downstream component. Due to flow separation over the leading edge radius of the duct, flow direction could not be resolved by the five-hole pitot probe in the 70% radial vicinity (area within the dotted lines). However, the kiel probe, represented by the black arrows, sensed highly angled flow above the rotor and reversed flow below the rotor.
Figure 4-20 (a) and (b) presents the flow angle $\theta$ above and below the rotor at a higher advance ratio of 0.21 (4000 RPM, $V_{WT}=10.7\text{m/sec}$). All points presented were measured at approximately the 70% radial location. Comparing Figure 4-20 (a) to Figure 4-18 (a) demonstrates that the higher advance ratio causes the flow angle $\theta$ to become larger. The rotor isn’t creating a large enough pressure differential to overcome the flow’s momentum and, therefore, is less effective in turning the flow into the duct. In (b), the flow angle $\theta$ is seen to increase slightly with the increase in advance ratio. Unfortunately, the increased advance ratio created a large amount of separation over the leading edge radius of the forward duct and the flow angles in many of the locations were unable to be resolved with the five-hold pitot probe. This leaves the flow angle between $90^\circ$ and $270^\circ$ unknown.
Figure 4-20: Downstream Flow Angle, $\theta$, within the Forward Fan as a function of Azimuthal Angle at an Advance Ratio of 0.21: (a) Above the Rotor, (b) Below the Rotor

The flow angle $\theta$ along the centerline of the model associated with the advance ratio 0.21 (4000 RPM, $V_{WT}$=10.7m/sec) is illustrated in Figure 4-21. As the advance ratio increases, so does the area of separation that the five-hole pitot probe can’t resolve and the downstream angle of flow through the duct. The front 70% and 30% radial locations, area within the dotted lines, can no longer be resolved by the five-hole pitot probe. The kiel probe, represented by the black arrows, measured highly angled flow above the rotor and reverse flow in below the rotor. In the rear of the fan, the downstream angle of the flow is lessened by passing through the rotor.

Figure 4-21: Downstream Flow Angle within the Forward Fan Along the Centerline of the Model at an Advance Ratio of 0.21
4.2.3 Force and Moment Measurements

The dual-ducted fan was mounted to the APB force and moment balance and placed inside the APB wind tunnel. The forces and moments associated with the model while in forward flight were collected while operating at various RPMs, forward flight speeds, and model angles of attack. Figure 4-22 to Figure 4-24 display, for angles of attack 0°, -5°, and -10° respectively, the dimensional total lift and non-dimensional lift coefficient $C_{L,TIP}$ explained in Equation 4-2, where ‘A’ is the sum of the disk area of both fans. Keep in mind that this system has no way of separating the rotor thrust from fuselage lift. Therefore, ‘Total Lift’ is defined by the sum of all the forces in the positive lift direction. The ‘Body Force’ represents the forces measured when the rotors are stationary. This is the closest approach to separating the forces, however, flow changes around the fuselage once the rotors are turned on, which will change the forces and moments experienced by the fuselage.

$$C_{L,TIP} = \frac{\text{Total Lift}}{\frac{1}{2} \rho AV_{TIP}^2}$$ Equation 4-2

Closely examining Figure 4-22 to Figure 4-24 reveals the changes in lift caused by altering the RPM, forward flight speed, or angle of attack. The most noticeable gain in lift is seen with the increase in the rotor speed. As the rotor speed is increased, the advance ratio is decrease and the pressure differential created by the rotor becomes larger, which improves the efficiency of turning the flow into the duct.

Increasing the wind tunnel speed, or forward flight velocity, also increases the lift produced by the model. At low forward flight speeds, the lift produced is very similar to a hover condition. This can be seen by the how flat the dimensional lift curve is at low wind tunnel velocities, or the flatness of the non-dimensional lift curve at low advance ratio. As forward flight speed is increased, the lift begins to increase exponentially. This is caused by the increased inflow velocities into the duct, which increases the mass flow resulting in increased thrust. For a
lifting body such as the fuselage, increase forward flight velocity will also result in increased lift. The greatest $C_{L,TIP}$ actually occurs at the highest advance ratio, which corresponds to 6000 RPM at 19 m/sec (65 ft/sec).

As the angle of attack of the model is decreased from $0^\circ$ to $-10^\circ$, the lift is seen to decrease. The thrust vector is being rotated from acting completely in the direction of lift to only have a component of the thrust adding to lift. As the angle of attack is decreased, the other component of the thrust will counteract the drag of the model. The non-dimensional lift shows that the angle of attack has the largest affect at higher advance ratios. Almost no change in the lift coefficient was measured at the lowest advance ratios. At the highest advance ratio, the lift coefficient decreased 6% when the angle of attack was changed from $0^\circ$ to $-5^\circ$ and 12% going from $-5^\circ$ to $-10^\circ$.

![Figure 4-22: Dimensional and Non-Dimensional Total Lift at $0^\circ$ Angle of Attack](image)
Figure 4-23 to Figure 4-27 display, for angles of attack 0°, -5°, and -10° respectively, the dimensional net drag and non-dimensional drag coefficient $C_{D_{\text{TIP}}}$ explained in Equation 4-3. The 'Net Drag' is defined by the sum of all the forces in the downstream direction, which is comprised of the model drag minus any component of the thrust vector that is upstream.

\[ C_{D_{\text{TIP}}} = \frac{\text{Net Drag}}{\frac{1}{2} \rho AV_{TIP}^2} \]  

Equation 4-3
Changes in drag caused by varying the rotor speed, forward flight speed, or angle of attack are shown in Figure 4-25 to Figure 4-27. Contrary to lift, the forward flight speed has the largest effect on the drag of the model. This is due to the force referred to as momentum drag by Kelly or ram drag by Fleming et al.\cite{18,19} This momentum drag is only experienced in forward flight because of the force required to turn and redirect the momentum of the free-stream velocity through the ducts. When in a static hover, flow converges into the ducts from all directions and results in a net sum of zero x-momentum. Once in forward flight, flow only travels downstream and, as explained in Equation 4-4, results in negative x-momentum. Fleming et al claim that this can account for 95% of the total drag experienced by an isolated duct at low velocities.\cite{19}

\[
\text{Drag} = -F_x = m_{\text{duct}} V_x = \rho A_e V_e V_x
\]

Equation 4-4

The fuselage of the dual-ducted fan will also experience large amounts of drag. One of these sources of drag can be attributed to pressure drag. The blunt leading face of the forward duct will experience a high stagnation pressure while in forward flight due to its shape. Just downstream, the retreating face most likely experiences low pressures due to the separated flow. This will result in an increasing drag force as forward flight increases. The same also occurs on the center-body and the rear-most part of the aft fan. Flow angles measured by the five-hole pitot probe show that flow travelling through the forward duct is angled. This would result in a stagnation pressure on the leading edge of the center-body. If flow is angled through the aft duct, which it most likely is according to the flow visualization images, a stagnation pressure will also be experience on the aft duct. The fuselage will also experience parasitic drag, or skin friction drag, as any moving vehicle would.

Decreasing the angle of attack from 0° to -10° was shown to decrease the total lift of the model. This was due to a component of the thrust vector being used to counteract the drag. If the drag reduction is large enough, angling the fuselage at high angle of attacks could be used to reduce the drag force that must be overcome by horizontal thrusters. The non-dimensional drag
shows that the angle of attack has a small effect over the entire range. At the highest advance ratio, the drag coefficient was constant when the angle of attack was changed from 0° to -5° and decreased 6% going from -5° to -10°.

Figure 4-25: Dimensional and Non-Dimensional Net Drag at 0° Angle of Attack

Figure 4-26: Dimensional and Non-Dimensional Net Drag at -5° Angle of Attack
In order to compare the lift and drag and determine an optimal configuration for flight, non-dimensional lift and drag are presented versus advance ratio in Figure 4-28 for various angles of attack. First, it is important to point out that lift is almost always greater than or equal to drag. However, the drag coefficient is increasing faster than the lift coefficient as advance ratio increases. As was discussed earlier, the angle of attack has little effect on the lift coefficient at low advance ratio, but this is the point it has the most effect on the drag coefficient. As the advance ratio increases, the higher angles of attack produce more lift and have little effect on the drag. The optimal range, greatest difference between lift and drag, is at the lowest advance ratios, but this is undesirable because the vehicle would have to travel at such low forward flight speeds. The drag must be reduced in order to design a practical vehicle. One attempt was a streamlined nose design, which will be discussed in Chapter 6 of this thesis.
Since the drag will increase with an increase in lift, the better way to analyze the drag of the vehicle with a constant lift. In addition, a full-scale vehicle will operate at a constant lift value while in steady, level flight. The power that is available to generate the excess lift could instead be used to oppose the drag, either in the form of transferring the excess power to auxiliary horizontal thrusters or changes the thrust angle. The wind tunnel model does not support horizontal thrusters; therefore, the thrust angle must be changed by changing the fuselage angle of attack. In order to do this, two RPMs were selected, 6000 and 8000 RPM, and the angle of attack was varied. The values of net drag were then extrapolated for two different values of lift, 16.6 N and 26.2 N, and are displayed in Figure 4-29 (a) and (b), respectively. It was found that the minimum amount of drag the vehicle would experience was at an angle of attack of -5° for both amounts of lift. This would relate to the angle at which the least amount of power is required in forward flight. However, this is also the slowest speed the vehicle would travel. Also, when the vehicle is at approximately a -11° angle of attack, the same amount of power would be required.
as when the vehicle is at a 0° angle of attack, but the vehicle would be able to travel 39% faster. This indicates that pitching the vehicle in order to use the lift fans to help produce thrust can be beneficial.

Figure 4-29: Net Drag of Model while Maintaining Constant Lift by Changing Angle of Attack: (a) Lift = 16.6 N, (b) Lift = 26.2 N

Many single ducted fan experiments have shown nose-up pitching moment while in forward flight. This is due to low pressure point on the leading edge radius caused by the flow separation. Fleming also attributes the problem to the ducted fan experiencing a center of pressure that is located above the duct. The reasoning for this goes back to the drag force referred to by Fleming as “ram drag”. The ram drag can be summed to act in the vertical center of the flow being turned into the ducts which results in a nose-up pitching moment. An example is shown in Figure 4-30. Since the center of pressure is above the fuselage, a moment arm is created and the drag also produces a nose-up pitching moment.

Figure 4-30: “Ram Drag” Resulting in a Nose-Up Pitching Moment
Figure 4-31 presents the dimensional and non-dimensional pitching moment measured from the dual-ducted fan in level forward flight at various rotor speeds. The pitching moment was non-dimensionalized using Equation 4-5, where $A$ is the sum of the area of both ducts and $R_m$ is the moment arm distance from the lateral axis that the model supports connecting to the balance act through to the hub of the forward rotor. The dimensional pitching moment as a function of wind tunnel speed displays that pitching moment increases due to increases in both rotor speed and wind tunnel velocity. The pitching moment coefficient as a function of advance ratio also summarizes this trend.

$$C_{m_{TIP}} = \frac{Pitching \ Moment}{\frac{1}{2} \rho V_{TIP}^2 A R_m}$$  \hspace{1cm} \text{Equation 4-5}

Figure 4-31: Dimensional and Non-Dimensional Pitching Moment at 0° Angle of Attack

Several approaches could be taken to reduce the nose-up pitching moment experienced by the vehicle. Unique to the dual fan, it is possible to use differential thrust from the forward and aft rotors to counteract the moment, but this would then introduce a yaw moment due to the differential torques. Control vanes could be positioned above and/or below the forward and aft
ducts allowing for attitude control and vehicle trim. The pitching moment could also be countered by a horizontal stabilizer creating lift on the rear of the vehicle. Without excessively extending the length of the vehicle, this stabilizer would need to be located near the aft fan. Since the aft fan is turning and ingesting flow, the flow experienced by the horizontal stabilizer could prove to be very complex and would result in very challenging design issues.

To summarize the forces and moments experienced in forward flight by the dual-ducted fan model, the lift, drag, and pitching moment are displayed on the same graph versus angle of attack for two rotor speeds in Figure 4-32. The wind tunnel velocity was set at 19.8 m/sec (65 ft/sec). Decreasing the angle of attack from 0° to -15° causes the lift force to decrease, approximately 23% when operating at 8000 RPM. The net drag is also shown to decrease proportionally with lift as the angle of attack is decreased. This is due to rotating the thrust vector from parallel with the lift vector to -15° off the lift axis. This creates a component of thrust that counteracts the drag of the vehicle. The pitching moment is relatively constant with angle of attack.

Figure 4-32: Lift, Drag, and Pitching Moment for Various Angles of Attack at 19.8 m/s: (a) 6000 RPM, (b) 8000 RPM
Chapter 5

**BASELINE DUAL DUCTED FAN COMPUTATIONAL FLUID DYNAMICS**

Unfortunately time, money, and wind tunnel schedules do not allow that every dual-ducted fan design to be fabricated and tested. Computational fluid dynamics (CFD) has become a reliable tool and has helped reduce the design and wind tunnel time required for projects such as airfoil or aircraft design. At this time, the validated computational tools are limited for design and analysis of dual-ducted fan vehicles.\(^{21}\) This chapter summarizes the work performed by Dr. James Dreyer and Penn State Applied Research Lab (ARL) using two levels of modeling fidelity were designed and validated. The first simulations used are steady flow predictions using a momentum source to model rotor effects. The second, more complex model was an unsteady, fully-resolved simulation which placed a rotating mesh rotor within the duct. Most of the results were also presented at the 2010 International Powered Lift Conference.\(^{22}\)

The three-dimensional steady and unsteady flow predictions used OVER-REL, an unsteady Reynolds-Averaged Navier-Stokes (RANS) flow solver. OVER-REL is an ARL in-house code that has undergone development and validation through a wide variety of configurations for over 15 years. The code solves the pseudo-compressible form of the RANS equations. The integral forms of the governing equations are discretized for cell-centered finite volumes on hybrid block-structured and overset meshes of hexahedral cells.\(^{23}\) The steady and dynamic (rotating rotors) overset grid communication is handled by the mesh processor SUGGAR.

Since the wind tunnel model was designed by hand, there was no computer-aided design (CAD) model to use in the generation of the fuselage mesh. Using photographic and measurement techniques, a geometrically similar CAD design was developed. The design is
presented in Figure 5-1. This was used for the generation of the fuselage mesh. A comparison of the CAD model to the physical model can be found in Appendix C.

Initial wind tunnel tests of the dual-ducted fan occurred within the Hammond wind tunnel. These are the wind tunnel tests that will be used to validate the CFD. In order to do so, the CFD must be run under the same conditions and within the same control volume. The dual-ducted fan fuselage was placed within a mesh design of the Hammond wind tunnel test section to maintain the wall effects that the model experiences within the test section.
5.1 Momentum Source

The computational mesh generated for the momentum source using the fuselage CAD file, Figure 5-3, was constructed using a hybrid block-structured and overset topology. It is made up of 239 blocks, over 5 million hexahedral cells, and over 5.5 million points. The dark, horizontal mesh pointed out in Figure 5-3 (c) are the force blocks where the momentum sources will act in place of the rotors. Before starting a simulation, a distribution and magnitude must first be set for the momentum source. For the initial momentum source simulations, a constant force per unit volume was distributed within the rotor sweep volume, shown in Figure 5-3 (c). This results in a circumferentially and radially uniform vertical thrust force for the forward and aft rotors. In hover, this is likely a reasonable approximation. In forward flight, as will be shown, this approximation breaks down. The thrust magnitudes are obtained from the experimental thrust measurements made in hover at the various RPMs shown previously in Figure 4-10. Cylindrical bodies are sized and placed in the center of each duct to represent the motor casing and propeller hubs. Unfortunately, one item left out of this design was the longitudinal struts that were discussed in Chapter 4 to possibly reduce the swirl within the ducts. For the momentum source CFD, this is irrelevant since the momentum sources only act in the axial direction.
Due to the simplicity of the momentum source approach, the time to complete a simulation of an operation condition is short. A variety of test cases can be performed. One of these cases is presented in Figure 5-4, where the model is operating in level flight at a forward flight velocity of 18.3 m/sec (60 ft/sec) and a rotor speed of 6000 RPM. Static pressures on the fuselage surface are displayed in (a). Red corresponds to a high pressure coefficient of 1.2, while blue is the lowest at -3. The capability of visualizing this static pressure distribution makes it easy to understand the high drag associated with these vehicles. High static pressure is calculated on all of the leading face of the front duct and the retreating interior face of the aft duct. A portion of the retreating interior face of the forward duct also shows an area of high static pressure. Although there isn’t a view provided, opposite the faces with high static pressure will be areas of low state pressure resulting in large amounts of fuselage pressure drag.
Figure 5-4 (b) presents a contour plot of the downstream velocity on a horizontal plane coincident with the plane of the rotors. Blue corresponds to reversed flow, green represents zero velocity, and red is the maximum downstream velocity. Most of the forward duct displays negative downstream velocities indicating stalled flow within the duct. The front portion of the aft duct contains similar reversed flow velocities. Figure 5-5 helps illustrate the reason for the downstream velocities acting opposite the expected direction. The figure separates flow through the two ducts using fluid particle paths. Red lines represent the particle paths of flow only through the forward duct. Green lines represent the particle paths flowing through the aft duct. The forward duct experiences large amounts of flow ingestion from below the rotor and a large amount of recirculation. The aft fan also displays a large amount of recirculation in the front half of the duct. These are the same areas displayed in Figure 5-4 (b) that had negative downstream velocities.
In principle, the momentum source simulation can be run as a steady simulation. These types of simulations are potentially useful for predicting the time-mean performance. However, as suggested by Figure 5-5, forward flight conditions result in flow separation and low frequency oscillations in the flow. A steady code is unable to evaluate these flow conditions. A time-accurate simulation is required to properly resolve the flow separation and oscillations.

Initial results from the momentum source model CFD were compared to measurements made within the Hammond wind tunnel. The model was placed in steady level flight at velocities of 3.0 m/sec (6.5 ft/sec) to 19.8 m/sec (65 ft/sec). Measurements were also obtained at two different rotor speeds, 6000 and 8000 RPM. Figure 5-6 (a) displays the measured (experimental) and the predicted (CFD) total lift as a function of wind tunnel speed. As mentioned before, lift is the sum of the vertical force produced by the rotors and the fuselage. At low forward flight speeds, the momentum source CFD exhibits good agreement with the experimental values. It was found in Chapter 4, that at low forward flight speeds, the total lift was very similar to the thrust measured in hover. Since the momentum sources were initialized with the thrust values obtained from the experimental hover cases, it can be expected that the low flight speed cases would be
similar. As the forward flight velocity is increased, however, the predicted values from the CFD significantly under-predict the measured values.

Figure 5-6 (b) displays the measured and the predicted net drag as a function of wind tunnel speed. The net drag is the sum of the horizontal force produced by the rotors and the fuselage. The momentum source CFD under-predicts the net drag by approximately 30% for both RPM cases and all forward speeds. The drag and most of the lift was unable to be accurately predicted by the momentum source CFD in these forward flight conditions. What is needed is a more sophisticated rotor force model that relates the spatial distribution of thrust to the state of the rotor onset flow, which will be discussed as an area of future work. Another method would be to computationally resolve the moving rotors, which is discussed in the next section.

5.2 FULLY RESOLVED ROTATING MESH

A step higher level of computational fidelity is to resolve the rotating rotors. The same fuselage mesh from the momentum source model is reused for the fully-resolved model. A CAD
design was created for the Master Airscrew 10x7 propellers used for the model by J. E. Eaton at Penn State ARL. A mesh could then be produced for the rotor and incorporated with the fuselage mesh, Figure 5-7. The fully-resolved mesh is now made up of 309 blocks, over 12 million hexahedral cells, and over 13 million points. This mesh becomes much more complicated than the momentum source mesh because the rotors are now rotating while the fuselage is stationary. Since the overset communication software, SUGGAR, is directly coupled with OVER-REL, it makes solving this dynamic overset problem possible. The inputs required for a fully-resolved simulation are the wind tunnel speed and the forward and aft rotor speeds, which can differ. In these simulations, there are no assumptions made as in the momentum source model. All of the thrust (and torque) is provided by the rotating rotor model.

![Figure 5-7: Hybrid Block-Structured and Overset Mesh for the Resolved Rotor Simulations: (a) Surface Mesh, (b) Slice Through the Horizontal Plane, (c) Detail of Aft Rotor Blade Showing Tip Gap [309 Blocks, 12.4M Cells, 13.5M Points]](image-url)
The computational model must now resolve the rotors spinning at thousands of revolutions per minute. The time resolution required for the time-accurate performance is significantly greater than that required by the momentum source simulations. As a result, the primary drawback of the fully-resolved CFD code is the run time required by flow solution to become statistically stationary. The total computational time required was greater by roughly an order of magnitude over the momentum source CFD.

The available run-time allowed for only one test case to be computed by the fully-resolved CFD; 6000 RPM at a forward velocity of 18.3 m/sec (60 ft/sec). However, once complete, the results are much more comprehensive. The proof of this is in the comparison of Figure 5-4 to Figure 5-8, which displays the static surface pressure contours obtained from the fully-resolved simulation. Clearly, the range of static pressures on the resolved rotors is significantly greater than on the fuselage. The low and high pressures on the upper and lower surfaces are clearly evident. The affect of low pressure tip vortices are visible on the interior surfaces of the duct along the rotor plane. The static pressure on the lower surface of the fuselage is greater than on the upper surface resulting in the fuselage acting as a lifting body.

Possible explanations for the observed experimental nose-up pitching moment are also made evident. On the forward duct, the upper leading edge radius exhibits low surface pressure because of the flow separation. The lower leading edge radius displays a much higher surface pressure, resulting in a lift force at the most forward point on the vehicle. The upper retreating edge radius on the aft duct also exhibits high surface pressures, while the lower retreating edge experience very low pressure resulting in a negative lift force at the aft-most location on the vehicle.
A time history for the lift forces calculated by the fully-resolved CFD is displayed in Figure 5-9. Each major gridline on the “time” axis denotes one full rotor revolution. The red symbols are the lift force generated by the fuselage; the blue are the thrust produced by the forward rotor; while the green are the thrust of the aft rotor. The computation displays large amounts of fluctuation in the initial three rotor revolutions. This can be contributed to the start-up transients and are not considered physically accurate.

After start-up, both rotors exhibit smaller, more consistent oscillatory loads at three peaks per revolution, and the fuselage experiences an oscillatory load at an approximate rate of one every other rotor revolution. Both of these oscillations can be attributed to a physical occurrence, which the fully-resolved solution provides insight into the causes. The high frequency and low amplitude oscillation observed in the rotor thrust curves is linked to non-uniformities in the inflow and outflow of the rotors. The lower frequency and larger amplitude fuselage oscillations can be attributed to shedding behavior off parts of the fuselage. The previous figure suggested some of the nose-up pitching moment is cause by pressure differentials on the forward and aft duct. The calculated lift force also predicts the thrust force of the forward fan to be greater than the aft fan adding to the nose-up pitching moment.
Figure 5-9: Lift Force History from the Fully-Resolved Rotor Simulation at $V_{WT}=18.3$ m/sec, 6000 RPM, and an Angle of Attack of $0^\circ$ [Red = Fuselage, Green = Aft Rotor, Blue = Forward Rotor]

The total lift and net drag predicted by the fully-resolved CFD is displayed in Figure 5-10 along with the experimental values. Unfortunately, the time demanded by the fully resolved only allowed for a single case to predicted, 6000 RPM at 18.3 m/sec (65 ft/sec). A favorable agreement for both the lift and drag was found between the CFD and the experimental. The total lift was over-predicted by approximately 5%. The net drag was under-predicted by approximately 3%.

Figure 5-10: Comparison of Fully-Resolved CFD Forces to the Experimental Force Results: (a) Total Lift, (b) Net Drag
While both the total lift and net drag predicted are within an acceptable error when compared to the experimental forces, more than one case should be calculated in order to validate the code. Since velocity magnitudes and angles have been experimentally measured within the forward duct, those velocities and angles could be extracted from the CFD to further validate the code instead. Currently, the cost to operate this code also limits its practical uses as an everyday design and analysis code. However, for providing insight into complex flow pathologies for these types of vehicles, it is a quite powerful given sufficient computational resources.
Chapter 6

COMPARISON OF THE STREAMLINED MODEL TO THE BASELINE

The addition of the duct to a rotor has several advantages, but two serious disadvantages develop when it must travel edgewise in forward flight, as was revealed in Chapter 4. High speed forward flight with a ducted fan is limited by excessive drag and separated inflow into the forward fan. The Bell/Urban Aero X-Hawk used vents on the leading face of the forward fan and trailing face the aft fan that would open during high speed travel and allegedly resolved this problem. It is not known that this has undergone validation yet. This will most likely require cyclic pitch to be incorporated into the forward rotor. Other recent research efforts investigated designs to reduce the separation of flow over the leading edge radius of a single duct fan, but little attention has been paid to a dual ducted fan vehicle.\(^{(24),(25)}\) There have also been limited efforts to reduce the overall drag force of a ducted fan vehicle while in edgewise flight. The streamlined nose, introduced in Section 2 of Chapter 2, was designed for the purpose of reducing both overall drag and inlet lip separation.

The purpose of the streamlined nose is two-fold. The first function of the streamlined design was to reduce the stagnation pressure on the leading face of the duct. A blunt shape such as the leading face of the forward duct on the baseline vehicle was expected to experience high drag. The surface static pressures calculated by the CFD, Figure 5-8, confirmed that the entire duct face was a high pressure region. The streamlined nose would result in a contour that would alleviate a portion of the high stagnation pressure.

The other anticipated benefit of the streamlined nose was to reduce the separation of flow over the leading edge radius of the forward duct thereby increasing the efficiency of the forward fan. The blunt nose of the baseline vehicle requires flow to travel up the leading face of the duct and then turn \(180^\circ\) in a short distance to remain attached to the fuselage and pass through the
rotor. Flow visualization and flow measurements in Chapter 4 demonstrated that the flow was unable to accomplish this and separated rendering portions of the fan ineffective. The streamlined nose reduced the turn required for the flow and was expected to reduce or eliminate the size of separation bubble over the leading edge radius.

### 6.1 Flow Visualization

As with the baseline mode, qualitative data about the flow around the model in forward flight was first obtained using the same flow visualization techniques. A vertical wire was placed in-plane with the vertical plane of symmetry. Advance ratios of 0.08 and 0.12 were observed by setting the rotor speed at 2000 RPM and the wind tunnel velocity at 2.0 m/sec (6.5 ft/sec) and 3.0 m/sec (10 ft/sec), which are presented in Figure 6-1. Separation of flow over the leading edge radius of the forward duct still occurred with the addition of the streamlined nose. It is difficult to detect any difference in the streamlines when comparing the flow visualization of the baseline model in Figure 4-14. The size of the separation bubble appears to be slightly smaller in Figure 6-1(a) than it was at an advance ratio of 0.08 on the baseline model.
6.2 FLOW MEASUREMENTS

Penetrating the forward duct of the vehicle and obtaining quantitative flow measurements with the kiel probe and five-hole pitot probe was the only method that would provide the results capable of determining any affect the streamlined nose had on the flow within the forward duct. The comparison of the velocity magnitudes measured by the kiel probe and five-hole pitot probe within the forward duct of the streamlined model are shown in Figure 6-2 as a function of azimuthal angle. Two wind tunnel speeds, 6.1 m/sec and 10.7 m/sec, are examined along with two rotor speeds, 4000 RPM and 6000 RPM. The kiel probe is shown with the standard symbols with the line, while the five-hole probe values are shown as the symbol with the bold, black outline. The agreement of the measurements made by the two probes is better than the comparison from the baseline model. The percent difference was typically less than 5%, with only a few locations reaching a difference over 10%.
Figure 6-2: Velocity Magnitudes within the Forward Duct of the Streamlined Model Measured by the Kiel Probe and the Five-Hole Pitot Probe: (a) Above the Rotor at $V_{WT} = 6.1$ m/s ($\mu=0.08$, $\mu=0.14$) (b) Above the Rotor at $V_{WT} = 10.7$ m/s ($\mu=0.12$, $\mu=0.21$) (c) Below the Rotor at $V_{WT} = 6.1$ m/s ($\mu=0.08$, $\mu=0.14$) (d) Below the Rotor at $V_{WT} = 10.7$ m/s ($\mu=0.12$, $\mu=0.21$)

Figure 6-3 and Figure 6-4 display the comparison of velocity magnitudes measured above and below the rotor within the forward duct as a function of azimuthal angle for the baseline model and the streamlined model in several different configurations. All the measurements presented were obtained at approximately 70% of the rotor radius. Rotor speeds of 4000 and 6000 RPM were tested at forward flight velocities of 6.1 m/sec (20 ft/sec) and 10.7 m/sec (35 ft/sec). The 4000 RPM cases are displayed in green, while the 6000 RPM cases are red. Solid lines and closed symbols represent the streamlined model, and the broken lines and open symbols
represent the baseline model. The average of the two probes is presented when measurements were able to be obtained with both.

The inflow velocities, velocities above the rotor, are shown in Figure 6-4 (a) and (b) at forward flight speeds of 6.1 m/s and 10.7 m/s respectively. The azimuthal area located around 180° was closely examined. This area was where the flow separation was measured within the baseline model and where the streamlined nose is expected to have the most effect on the flow. When the rotor is set at 6000 RPM, the inflow velocity of the streamlined model at 180° is approximately 40% higher than the baseline model. The two surrounding measurements also displayed higher velocities for the streamlined model. When at 4000 RPM, the difference is only noticeable while the wind tunnel is at 6.1 m/s with an inflow velocity gain around 16%. When at 10.7 m/s and 4000 RPM, the streamlined nose has little to no effect on the inflow velocity at 180°. For all the test conditions, most of the measurements made further downstream of the inlet lip exhibits slightly higher velocity magnitudes.

Figure 6-3: Comparison of the Averaged Velocity Magnitudes from the Kiel Probe and the Five-Hole Pitot Probe for the Baseline and Streamlined Model Above the Rotor at 70% Radius: (a) \( \text{V}_{\text{WT}} = 6.1 \text{ m/sec} \) (\( \mu = 0.08, \mu = 0.14 \)) and (b) \( \text{V}_{\text{WT}} = 10.7 \text{ m/sec} \) (\( \mu = 0.12, \mu = 0.21 \))
Although the streamlined nose may not have completely eliminated the separation over the front lip as hoped, Figure 6-4 demonstrated that it did make a difference in improving the inflow velocities overall and especially at 180°. The test case at the lowest advance ratio, 6000 RPM at 6.1 m/sec ($\mu=0.08$), appears to have little inlet lip separation. Although, images from the flow visualization displayed streamlines separating from the leading edge lip.

The only test case that didn’t show improvement at 180° was the case with the highest advance ratio, 4000 RPM at 10.7 m/sec ($\mu=0.21$). The argument can be made that 4000 RPM may be an inefficient speed for these rotors. At 6000 RPM and 10.7 m/s ($\mu=0.14$), the improvement in inflow velocity was greater than at 4000 RPM and 6.1 m/s ($\mu=0.12$). The lower advance ratio would be expected to exhibit a larger improvement. Therefore, 6000 RPM may be a more optimal RPM for these rotors and higher RPMs should be explored.

The comparison of the outflow velocities for the two model configurations are exhibited in Figure 6-4. The inflow velocities suggested reduced flow separation with the addition of the streamlined nose, but in examining the outflow velocities, no notable gains are measured in the area of interest, the azimuthal 180° location and nearby points. Reverse flow was still experienced between the azimuthal locations of 152° and 208°. Some increases in outflow were measured at locations further downstream of the leading edge radius. Although the streamlined nose did little to affect the outflow velocities closest to the leading edge of the duct, the overall velocities above and below the rotor appeared to be greater than the baseline model. If this is true, it would result in increased mass flow through the duct, which would translate into increased fan thrust (and lift).
6.3 FORCE AND MOMENT MEASUREMENTS

Not only was the purpose of the streamlined nose to reduce flow separation off the leading edge radius of the forward duct, but it was also meant to reduce the drag produced by the vehicle. The model, with streamlined nose attached, was mounted inside the APB wind tunnel on the APB balance so force and moment measurements could be obtained at various wind tunnel speeds, rotor speeds, and angles of attack. The total lift, net drag, and pitching moment were then compared to measurements obtained from the baseline model in the subsequent figures. The streamline model is represented in the figures with solid lines and solid symbols, while the baseline is represented by broken lines and open symbols.

Figure 6-5 to Figure 6-7 present the total lift force of both models for angles of attack of 0°, -5°, and -10° respectively. On the left, the dimensional lift is displayed in Newtons as a
function of the wind tunnel velocity in m/sec. On the right, the non-dimensional lift coefficient, \( C_{L, \text{tip}} \), is presented as a function of the advance ratio. Small gains and losses in lift are associated with the addition of the streamlined nose to the dual-ducted fan model. Most of the dimensional and non-dimensional was found to be very similar. Although the flow measurement suggested there may be increased rotor thrust due to higher flow-through velocities, the force balance was unable to measure a change in most of the configurations. The largest differential measured was at the highest wind tunnel velocity, 19.8 m/sec (65 ft/sec) and -10° angle of attack.

![Figure 6-5: Dimensional and Non-Dimensional Lift for the Streamlined and Baseline Model at 0° Angle of Attack](image-url)
Figure 6-6: Dimensional and Non-Dimensional Lift for the Streamlined and Baseline Model at -5° Angle of Attack

Figure 6-7: Dimensional and Non-Dimensional Lift for the Streamlined and Baseline Model at -10° Angle of Attack
Figure 6-8 to Figure 6-10 compare the net drag force of the two model configurations at angles of attack of 0°, -5°, and -10°. The dimensional net drag is presented on the left as a function of wind tunnel velocity, and the non-dimensional drag coefficient, $C_{D,TIP}$, is on the right as a function of advance ratio. It was expected that the streamlined nose would reduce the high stagnation pressure experienced by the blunt shape of the baseline dual-ducted fan and reduce the drag experienced by the model. The experiments performed in APB demonstrated that the streamlined nose had a small effect on the drag. It should be kept in mind that this was only simple proof of concept design. Integration of the streamlined nose to the baseline model was not very smooth, as seen in Figure 2-11, and would introduce additional drag.

Table 6-1 to Table 6-3 displays the percent difference in net drag measured between the two models at the three angles of attack. At low wind tunnel velocities, the baseline and streamline data are nearly indistinguishable from each other in the figures, but in the percent difference calculated in the tables, small increases and decreases are seen. Increasing the wind tunnel speed to 19.8 m/sec (65 ft/sec) causes a noticeable difference between the two models in the figures, although the tables still calculate small percent differences. While at an angle of attack of 0°, the streamlined model unexpectedly experienced equal or greater drag than the baseline model for most cases. When the angle of attack was decreased to -5° or -10°, the drag of the streamlined model was normally slightly less than the baseline model.

The largest drag differences were measured when the rotors were off, the body force measurements. In this case, flow is not being turned into the ducts so there is no momentum drag. These cases show the effect of the streamlined nose has on reducing the pressure drag. Once the rotors are rotating, the momentum drag becomes the dominating component of drag. The streamlined nose has no effect on reducing the momentum drag.
Figure 6-8: Dimensional and Non-Dimensional Drag for the Streamlined and Baseline Model at 0° Angle of Attack

Table 6-1: Percent Difference in Drag between the Streamlined Model and Baseline Model at a Fuselage Angle of Attack of 0° (Negative refers to a drag decrease due to the Streamlined Nose)

<table>
<thead>
<tr>
<th>Wind Tunnel Velocity [m/sec]</th>
<th>Rotor Speed [RPM]</th>
<th>Advance Ratio [$\mu$]</th>
<th>Percent Difference [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.0</td>
<td>0</td>
<td>--</td>
<td>-0.7</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.042</td>
<td>+4.4</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.031</td>
<td>-7.5</td>
</tr>
<tr>
<td>6.1</td>
<td>0</td>
<td>--</td>
<td>-9.0</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.050</td>
<td>+1.8</td>
</tr>
<tr>
<td>10.7</td>
<td>0</td>
<td>--</td>
<td>-26.7</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.146</td>
<td>+5.3</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.109</td>
<td>+2.2</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.087</td>
<td>-0.8</td>
</tr>
<tr>
<td>19.8</td>
<td>0</td>
<td>--</td>
<td>+2.3</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.270</td>
<td>+3.9</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.203</td>
<td>+6.2</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.162</td>
<td>-1.0</td>
</tr>
</tbody>
</table>
Figure 6-9: Dimensional and Non-Dimensional Drag for the Streamlined and Baseline Model at -5° Angle of Attack

Table 6-2: Percent Difference in Drag between the Streamlined Model and Baseline Model at a Fuselage Angle of Attack of -5° (Negative refers to a drag decrease due to the Streamlined Nose)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>0</td>
<td>--</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.050</td>
<td>+6.8</td>
</tr>
<tr>
<td>10.7</td>
<td>0</td>
<td>--</td>
<td>-14.5</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.146</td>
<td>+1.4</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.109</td>
<td>+0.2</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.087</td>
<td>-0.9</td>
</tr>
<tr>
<td>19.8</td>
<td>0</td>
<td>--</td>
<td>-14.9</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.270</td>
<td>-4.0</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.203</td>
<td>-1.5</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.162</td>
<td>-2.0</td>
</tr>
</tbody>
</table>
Figure 6-10: Dimensional and Non-Dimensional Drag for the Streamlined and Baseline Model at -10° Angle of Attack

Table 6-3: Percent Difference in Drag between the Streamlined Model and Baseline Model at a Fuselage Angle of Attack of -10° (Negative refers to a drag decrease due to the Streamlined Nose)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3.0</td>
<td>0</td>
<td>--</td>
<td>-4.1</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.042</td>
<td>+14.8</td>
</tr>
<tr>
<td>6.1</td>
<td>0</td>
<td>--</td>
<td>-10.5</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.050</td>
<td>+9.64</td>
</tr>
<tr>
<td>10.7</td>
<td>0</td>
<td>--</td>
<td>-3.3</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.146</td>
<td>-1.2</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.109</td>
<td>-1.3</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.087</td>
<td>-0.3</td>
</tr>
<tr>
<td>19.8</td>
<td>0</td>
<td>--</td>
<td>-8.48</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>0.270</td>
<td>-4.5</td>
</tr>
<tr>
<td></td>
<td>8000</td>
<td>0.203</td>
<td>+2.7</td>
</tr>
<tr>
<td></td>
<td>10000</td>
<td>0.162</td>
<td>-3.9</td>
</tr>
</tbody>
</table>
Figure 6-8 to Figure 6-12 compare the dimensional and non-dimensional pitching moment of the two model configurations at angles of attack of 0°, -5°, and -10°. If the streamlined nose reduced the separation of flow off the leading edge radius of the forward duct, the nose-up pitching moment should be reduced because of the increased static pressure on the leading edge radius. However, the streamlined nose may cause two changes that would also increase the pitching moment. Decreasing the flow separation off the forward duct would result in a more efficient forward rotor that would produce more thrust. More thrust from the forward duct results in a large nose-up pitching moment. Also, if the nose turns out to a lifting body, the lift produced at the front of the model will add to the nose-up pitching moment. The following figures portray the model with the streamlined nose to experience a greater nose-up pitching moment than the baseline model for most of the range tested. The only time the streamlined model exhibited similar or lower nose-up pitching moments was during operation at the highest wind tunnel velocity tested, 19.8 m/sec (65 ft/sec).

Figure 6-11: Dimensional and Non-Dimensional Pitching Moment for the Streamlined and Baseline Model at 0° Angle of Attack
Figure 6-12: Dimensional and Non-Dimensional Pitching Moment for the Streamlined and Baseline Model at -5° Angle of Attack

Figure 6-13: Dimensional and Non-Dimensional Pitching Moment for the Streamlined and Baseline Model at -10° Angle of Attack
The results of the model with the streamlined nose were disappointing once compared to the baseline model. It was anticipated that the streamlined design would reduce the separation of flow over the leading edge radius and into the forward fan by reducing the amount of turning expected of the flow. It was also expected that the streamlined nose would help reduce the drag of the vehicle by lessening the stagnation pressure on the blunt leading edge face of the baseline model. Although the redesign of the nose shape of the model was unsuccessful in meeting expectations, it provided a great deal of insight into the aerodynamic properties of the model.

The addition of the streamlined nose only improved the velocity magnitudes measured above the rotor. There was little change observed in the velocity magnitudes below the rotor. Since the streamlined nose didn’t entirely reduce the flow separation, two new issues are presented for the dual-ducted fan model. The first was that more attention may be required in the design of the leading edge radius into the forward fan. Graf and Martin both present experimental results on the effects of the size and shape of the leading edge radius on a ducted fan in static and forward flight.\(^{(24)}\),\(^{(20)}\) This radius wasn’t specifically shaped or sized during design and fabrication of the dual-ducted fan model. The second issue would be the possibility of the addition of more aggressive rotors. The use of higher performance rotors, ones with more pitch for example, would result in an increase pressure differential across the rotor due. This should lead to reduced flow separation over the lip. However, the more aggressive rotors would have higher power requirements. The trade-offs between the increase in power required and any increases due to the possible reduction in flow separation would need to be studied in the wind tunnel. Also according to Equation 4-4, higher performance rotors would increase the mass flow, thereby, increasing the momentum drag.

Even though the streamlined nose was generally ineffective at reducing the drag of the vehicle in forward flight, the drag is better understood now because of this experimental design. The stagnation pressure on the blunt face of the baseline model was thought to be a major
contributor to the overall drag of the vehicle. Since the streamlined nose had little effect on the drag, this appears to be an incorrect assumption. This narrows the major sources of drag down to the high stagnation pressures exhibited on the retreating faces of the forward and aft ducts in Figure 5-4 and the momentum drag, which was discussed in Chapter 4. Since the stagnation pressure on the blunt leading face of the baseline model was determined to have minimal affect on the drag, it can be assumed the drag associated with the stagnation pressures on the retreating faces of both ducts are also minimal and the major contributor is the momentum drag.

The forward vents on the X-Hawk, Figure 1-4, and the AirMule, Figure 1-5, appear to be a very good design after reviewing the results of the streamlined nose. Opening the forward vents while in high speed forward flight should result in decreased drag and a decrease nose-up pitching moment. With the vents open, there will be little stagnation pressure on the leading face of the duct, but more importantly, less flow will be required to turn into the duct. More flow will be entering through the forward vents, which should result in less momentum drag due to less flow turning into the duct and possibly a lower nose-up pitching moment due to reduce flow separation. The drawback to this design would be that, due to the rotor now experiencing forward flight like an isolated rotor, it would most likely require cyclic controls.
Chapter 7

CONCLUSIONS AND SUGGESTIONS FOR FUTURE WORK

7.1  SUMMARIZING CONCLUSIONS

The final remarks of this thesis will summarize the three chapters presenting results of the work performed on the dual-ducted fan and reiterate some important conclusions that were drawn from the effort. Ducted fans have been found to offer performance gains over an isolated rotor when experiencing a hover situation or axial flow. When the ducted fan is required to perform in an edge-wise flow, those performance gains are diminished by losses caused by the duct. The main objective of this research was to present a dual-ducted fan, a vehicle that has received little attention, and characterize its aerodynamic properties while stimulating more research pertaining to the vehicle to follow.

7.1.1  Hover and Forward Flight Experiments of a Baseline Dual-Ducted Fan Model

The baseline model is a dual-ducted fan wind tunnel model designed and built with an expected full-scale role as a UAV medevac. During the design, other dual-ducted fan designs were analyzed and applied, but the design remained generic as possible. The model was tested extensively in static conditions, as well as in forward flight. In both conditions, flow velocities and flow angles were measured within the forward duct. The thrust was measured in hover with the use of a force and moment balance. In forward flight, the lift, drag, and pitching moment were also measured and presented.

While in a static hover, flow velocities through the ducts were measured with a mini-vane anemometer and five-hole pitot probe. The mini-vane anemometer was placed at several
locations above and below the axis of symmetry of the forward and aft ducts. Above the forward duct, it was found that the velocity magnitudes varied span-wise and also experienced change when both fans were operating. The cause of the varied inflow was related to the effect the motor mounting struts may have on the flow and their absence in the front-most part of the forward fan. From the results, it also appears that the aft fan slightly deprives the forward fan of flow. The outflow velocities of the two fans were measured to be very similar. The largest difference is noticed in the front half of the forward fan when only the forward fan is operating. This could be related to the motor support struts again. They could act as stator vanes and increase the axial velocity for the rear half of the forward fan and the aft fan while their absence allows for more swirl in the front half of the forward fan.

The five-hole pitot probe was used to penetrate the forward duct and obtain velocities while in hover. The five-hole pitot probe allows for velocity components and flow angles to be calculated from an unknown flow. In the inflow and outflow of the forward fan, it was found that the axial velocity is the dominate velocity. There were measurable velocities in the downstream and cross-stream direction, which describe the contraction of the slipstream. The flow angles calculated with the five-hole probe also help illustrate the contraction of the slipstream and show, that in the outflow, the flow is almost purely axial.

The thrust and pitching moment of the model were measured via the APB balance. The total thrust, sum of the rotor thrust and duct thrust, is found to reach nearly 36 N at a maximum rotor speed of 10,000 RPM. This correlates to a maximum thrust coefficient of 0.022. The thrust coefficient of the dual-ducted fan is found to increase with RPM instead of remaining constant like an isolated rotor. The thrust increase with RPM can be related to increases in duct thrust due to increased outflow velocities. The pitching moment was found to be nearly zero for all rotor speeds except 10,000 RPM. A slight nose-down pitching moment is experience at this speed.
Forward flight introduced many more complexities to the flow around the dual-ducted fan. Flow visualized via a vertical smoke wire on the longitudinal axis of symmetry helped to provide qualitative insight into the flow characteristics. The smoke streamlines suggested separation of flow off the leading edge radius of the forward fan. They also suggested that under the rotor of front-most part of the forward fan experienced reverse flow, or ingested flow from the lower surface.

Flow velocity measurements obtained within the forward duct with a kiel probe and a five-hole pitot probe while the model was positioned at an angle of attack of -10° were able to describe the reverse flow, or the ingestion of flow, below the fan in the front half. The separation of flow was also confirmed with the decrease in velocity magnitude at 70% and 30% of the rotor radius as the front half was approached. As the advance ratio increased, the more the velocity magnitude decreased in the front half of the forward fan. Unfortunately, the test locations were too sparse to express the exact size of the separation bubble off the leading edge radius and couldn’t be compared directly to the flow visualization images.

Forward flight experiments were also performed with the use of the APB balance in order to measure forces and moments experienced by the model. The lift, drag, and pitching moments for various rotor speeds, wind tunnel velocities, and angles of attack were presented in this thesis. The total lift of the model, sum of rotor thrust and fuselage lift, was found to be highly dependent on the rotor speed. The largest gains in total lift were measured when the rotor speed was increased. When the wind tunnel velocity was increased to the maximum test case 19.8 m/sec (65 ft/sec), large increases in lift were also recorded. Then the model was operating in the two lower wind tunnel velocities, the total lift was similar to the total thrust measured in hover. As the angle of attack of the model was decreased from 0° to -10°, the lift was also seen to decrease due to a component of the rotor thrust beginning to act to counteract drag.
Three types of drag were presented that the model would experience in forward flight: parasite drag, pressure drag, and unique for ducted fans, momentum drag. Momentum drag is the result of having to turn and direct the momentum of the free-stream velocity down through the ducts. While the drag was typically less than the lift produced, these three types of drag resulted in a large force that would require significant amounts of power to overcome. It was also found that decreasing the angle of attack in order to gain a component of thrust in the direction to counteract drag proved to be counter-productive. The decrease in total lift of the model was greater than the resulting decrease in drag, especially at higher advance ratio.

The pitching moment measured by the APB balance established that the dual-ducted fan also experienced a nose-up pitching moment like other single-ducted fan experiments had exhibited. As rotor speed and wind tunnel velocity were increase, the nose-up pitching moment experienced were increased. As the advance ratio increases, the separation off the leading edge radius of the forward duct grows. The larger separation bubble results in a greater area on the upper surface forward duct experiencing lower pressure. Since the lower surface of the forward duct experiences high pressure, this results in a nose-up pitching moment that increases with an increase in flow separation. The fully-resolved CFD case also calculated differential thrust produced by the forward and aft rotors. The forward rotor was predicted to produce approximately 28% more thrust than the aft rotor, which would exaggerate the nose-up pitching moment. Several methods of dealing with the pitching moment were presented, such as differential operation between the two fans to counteract the moment or a horizontal stabilizer.

The baseline dual-ducted fan was found to be inefficient in its current design and must be improved upon in order to perform as a practical vehicle. Currently, the drag increases too rapidly with an increase in forward flight velocity. The separation of flow also is a disadvantage of the vehicle. It is one of the causes of a high nose-up pitching moment, which would be difficult to manage, and reduces the efficiency of the forward fan. Without a design that can
perform efficiently as higher forward flight speeds, the vehicle will be not be a viable vehicle for any kind of air transportation.

7.1.2 Design and Validation of CFD Codes for a Dual-Ducted Fan

Since wind tunnel testing is both timely and expensive, there is a need for a CFD code that can be used for the design and analysis of a dual-ducted fan. Working with the Penn State Applied Research Lab, two different CFD codes were developed and validated with the use of the baseline model. The first was a steady code which used momentum sources in place of the rotors in order to simulate the thrust force. The second code was unsteady and consisted of a time-dependent rotating mesh of the rotors.

The purpose of the momentum source code was to support the design phase using idealized rotor performance in rapid turn-around, steady RANS simulations. These simulations are characterized by computational meshes that do not resolve the rotor blades but model the rotor sweep volume instead. Using thrust data obtained from hover configurations of the dual-ducted fan model, the rotor sweep volumes are filled with a force per unit volume in order to represent estimated rotor thrust at various rotor speeds in forward flight conditions. This simple, low level modeling fidelity allowed for multiple simulations to be performed per day, which would be sufficient to support a design exercise. However, the accuracy of such a code is somewhat compromised for the turn-around speed. Results of the momentum source code are compared to the experimental results to assess the accuracy of the code and determine if it is a reliable design tool.

At low forward flight speeds, the lift and drag predicted by the CFD is within an acceptable accuracy when compared to the experimental values obtained in the wind tunnel. It was pointed out in Chapter 4 that the baseline model displayed only small gains due to forward
flight velocity for the first two velocities tested. Since the rotor forces for the momentum source model are based on measurements obtained from hover, it would be expected that the forces predicted by the CFD are similar. When the forward velocity is increased to the maximum tested experimentally, the code severely under-predicts the lift and the drag force of the model.

The other CFD model consisted of fully-resolved, rotating meshes of the rotors within the ducts. In these simulations, the rotor speed is chosen and the time step resolution for the unsteady RANS is selected so to accurately resolve the highest frequency oscillations. With the addition of the rotor meshes and the time-accurate nature of the simulations, the resource requirements of a run with this code is at least an order of magnitude larger than the momentum source code. Design iterations require simulations with fast turn-around times, so this model would be inappropriate in a design role. However, once validated, it could serve the important role of performance prediction once a detailed design is established.

The detailed results of the single case predicted by the fully-resolved CFD code displayed blade passage fluctuations in the rotor thrust forces and shedding behavior off the fuselage in the fuselage lift forces. The total lift force predicted was found to be over-predicted by only 5% when compared to the experimental values. The net drag was only under-predicted by 3%. Although these predictions are very close to experimental, further study is needed to validate the code. More cases should be performed to ensure it works for a full range of configurations. It would also be beneficial to compare flow field metrics, such as the flow velocity magnitudes above and below the forward rotor.

7.1.3 Forward Flight Experiments of a Streamlined Dual-Ducted Fan Model

After analyzing the baseline model, three major fallbacks to a dual-ducted fan vehicle were apparent in the forward flight test cases. First, the ratio of lift to drag of the model was
found to diminish quickly as forward flight speed increased. The amount of drag increased much
too rapidly with the increase in forward velocity. Also, flow was found to separate from the
leading edge radius of the forward duct when in forward flight. The size of this separation bubble
would also grow rapidly as forward velocity increase. This would result in last two undesirable
effects. With the flow separated from the leading edge radius, a low pressure region is left on the
upper surface of the forward duct. The lower surface of the forward duct experiences a much
higher pressure and a lift force is created at the most forward point on the model resulting in a
nose-up pitching moment. The separated flow could also results in a less effective forward fan.
As the separation bubble grows, the effective duct area decreases and so does the effective fan
area. With these issues, a full-size vehicle such as the dual-ducted fan model is impractical
because it would incapable of reaching effective forward flight speeds and still remain efficient.
The streamlined nose was designed with the drawbacks of the baseline model in mind.

If the blunt shape of the leading face on the forward duct was replaced with a more
streamlined shape, it was predicted that it would help reduce or eliminate the limiting factors of
the baseline model. The drag experienced by the baseline model should be reduced with the
addition of a streamlined nose because the stagnation pressure on the streamlined nose would be
much less. There was also the possibility that it would reduce or eliminate the separation of flow
over the leading edge radius. The blunt nose of the baseline model required the flow to turn
nearly 180° in a short distance in order to remain attached. With a streamlined nose, the flow
would only be expected to be turned approximately 90° by the fan in order to remain attached.

Design of the streamlined nose was accomplished with the use of SolidWorks. Using the
fuselage CAD file, the nose was designed to overlay the baseline model and attached on the sides
of the forward duct. After several design iterations, production of the model was handled through
rapid-prototyping.
Flow visualization was performed on the streamlined model with the use of a single vertical smoke wire located along the longitudinal axis of symmetry. The resulting images were very similar to those captured from the baseline model. The difference was indistinguishable until velocity magnitude measurements were obtained with the kiel probe and five-hole pitot probe. Increased velocity magnitudes were measured above the fan, although the increase depended on advance ratio. The greatest change was found at the lowest advance ratio, 0.08, which resulted in an increase of velocity by 40% when measuring closest to the forward leading edge radius. When the largest advance ratio was tested, 0.21, the velocity increase due to the addition of the streamlined nose had completely diminished. Unfortunately, measurements obtained below the rotor showed no change in velocity magnitudes when measuring closest to the leading edge radius. Since the flow through the duct is angled, the separation bubble grows as flow travels axially through the duct. Flow measurements were too sparse to record any improvement made in the outflow velocities by the streamlined nose.

Force and moment measurements were obtained from the streamlined model while in forward flight and compared to the baseline model. The lift force generated by the streamlined model was similar to the baseline model over the entire range of tests. This was expected since only small increases in velocity were measured within the forward duct.

The drag force associated with the streamlined model was also very similar to the baseline model. Only at the highest forward velocity tested, did a measurable variation begin to emerge. At an angle of attack of 0° and the highest forward velocity, the drag of the streamlined model was greater than the baseline model, but at angles of attack of -5° and -10°, the drag decreased slightly with the streamlined nose. The addition of streamlined nose would not have resulted in a change in momentum drag. Therefore, the momentum drag was found to be a large contributor to the overall drag experienced.
The addition of the streamlined nose also resulted in increased nose-up pitching moments to be measured for most of the test cases. Only at the higher advance ratios did the streamlined nose result in similar or decreased nose-up moments to be measured when compared to the baseline model.

The streamlined nose didn’t reduce the drag or flow separation as was hoped during design, but the unsuccessful design did allow more to be learned about the factors limiting the dual-ducted fan from increase forward flight velocities. Even though the overall drag was not decreased by the streamlined nose, it should have still decreased the drag associated with the stagnation pressure on the forward duct. It can now be assumed that this stagnation pressure only makes up a small portion of the overall drag. Therefore, it can also be assumed that the majority of the drag is related to the momentum drag, or the turning of the flow into the ducts.

Although increased velocities were measured above the fan with the streamlined nose, flow separation over the leading edge lip remained a serious problem. Other research suggests the size and shape of the leading edge radius requires more attention in order to reduce the separation. Higher performance fans that would create a larger pressure differential may also prove to assist in decreasing the flow separation.

### 7.2 Suggestions for Future Work

#### 7.2.1 Baseline Model

The baseline model should continue to be a heavily explored VTOL vehicle. Once the experimental wind tunnel model is better understood, it could be expanded to a full-sized vehicle and could prove to be a very valuable addition to commercial and military air vehicles. However, the results of the work presented in this thesis demonstrated that much is left to be learned about
these vehicles in order to make it more practical. New experiments can be performed to better quantify the flow around the vehicle. Also, new fuselage shapes can be designed and integrated into the baseline model (much like the streamlined nose was) in order to improve upon the forward flight performance.

One improvement in flow measurements would be to increase the resolution of flow measurements within the forward fan. This would not only allow for a comprehensive view of the flow through the duct, but it would also allow for better representation of the flow separation off the leading edge radius. This could be accomplished several different ways. One approach would be to drill more holes or cut a slot in the side of the forward duct to allow for more access locations for the five-hole pitot probe and kiel probe. Instead of cutting into the side of the forward duct more, it may be possible to locate the probes just above and below the duct and traverse them throughout the duct area. Once the experimental velocity measurements validate velocities measured in the fully-resolved CFD, it could also be used to create a full flow profile within the forward duct.

Currently, flow visualization images contain the only information describing flow through the aft duct. Using the same methods as the forward duct, locations on the side of the aft duct should be drilled for kiel probe and five-hole pitot probe access. Measurement locations within the aft duct should be limited to the current locations measured in the forward duct. If it is found that more complex flow exists, such as separation, the resolution should then be increased to help thoroughly classify the flow.

Separation of rotor thrust from fuselage lift would be beneficial in understanding the effects of the ducts and the effect flow separation has on rotor performance. The most simplistic approach would be to apply foil-type strain gauges to the longitudinal struts supporting the motors. They could measure strain in the support struts caused by the rotor thrust. Figure 7-1
displays possible locations of the gauges. Once attached, weights could be applied to the motors in order to simulate thrust and calibrate the strain gauges.

Figure 7-1: Possible Strain Gauge Locations for Measuring Rotor Thrust

The design of the Bell / Urban Aeronautics X-Hawk and the Urban Aeronautics AirMule both include a vented forward duct. They claim that, while in high speed flight, these vents are opened to reduce the drag and pitching moment. The only publicly available results that support this were presented by Myers. While it was a limited initial view of the concept, it was also performed on a single fan model. The forward vents should be adapted into the baseline model and thoroughly tested in order to determine their effectiveness in dual-ducted fan applications.

Two different approaches could be taken to integrate the forward vents into the dual-ducted fan. The first method would be to remove the front half of the forward duct completely and streamline the expose section of the duct, similar to Figure 7-2. This would present an ideal case of opened vents. However, the removal of the original front half of the duct involves much careful planning and execution in order to enable the model to be returned to its baseline configuration at any time.

Figure 7-2: Open Forward Duct Concept
Two suggested cut lines are displayed in Figure 7-3. The preferred cut, shown in (a) by the red lines, is straight through the surrounding fiberglass and foam, but angled across the PVC ring. The angled cut across the PVC provides a surface for the mounting of other shapes. The PVC is only 12.7 mm (0.5 inch) thick. The cut should be highly angled in order to provide enough surface area for mounting. Threaded inserts could be installed on the angled portion of the original model. Flat head bolts would then pass through any new front half duct shapes and attach the new shape by threading into the threaded insert. If more strength in the attachment is thought to be necessary, a keyway could be cut into the angle on the original and the key would protrude from any new front half shapes. The original front half of the duct would require material to be added to make up for the saw cut in order to reinstall it on the model.

(a) Angled Cut                                               (b) Straight Cut

Figure 7-3: Top View of Baseline Model Displaying Possible Cut Lines: (a) Angled Cut, (b) Straight Cut

The second cut, shown in Figure 7-3 (b), is a straight cut through the fiberglass, foam, and PVC ring. This is a much simpler cut to make but requires more design for an easy attachment. One possible way to attach the new front halves of the duct using this cut would be very similar to the design of a hinge. One half of the “hinge” would attach to face on the PVC
ring created by the cut. The other half would be attached to the face of the new front half being applied. When the new front half shape is mated to the original model, a pin is dropped in from the top to connect the two, exactly like the hinge of a door. This connection would occur on both sides of the model.

Once the original front half of the forward duct has been removed, new shapes can be designed in a CAD program, such as SolidWorks, and rapid-prototyped. The new shapes would include the rounded shapes to be placed on the exposed faces of the duct during the open duct experiments and a vented front duct half similar to the design of the X-Hawk or AirMule. Other shapes could possibly include a front half designed to easily accept different leading edge radius shapes or ducts with different chord lengths.

7.2.2 Computational Fluid Dynamics

The CFD effort for dual-ducted fans also has holds much potential for further investigation. The fully-resolved simulations should be further validated. This could be done by performing several other configurations of rotor speeds and wind tunnel speeds and comparing them to the experimental values of lift, drag, and pitching moment. Validation could also be done by extracting velocities through the forward duct from the simulations and comparing them to the values measured by the five-hole pitot probe and kiel probe.

Since the returns of momentum source simulations possessed error at higher forward flight velocities, there is still a demand for simulations that would allow for quick turn-around and could be used during design efforts. One possible solution could be a hybrid simulation that couples the steady momentum source model with a blade element code. The iterative coupling of the blade element code would help compensate for the non-uniform flow that the momentum
source code overlooks. This effort has recently been initiated by Jason Halwick and Jules Lindau.
REFERENCES


# APPENDIX A

## MATLAB CODE AND RESULTING CALIBRATION FILE FOR FIVE-HOLE PITOT PROBE CALIBRATION

### A.1 Raw Calibration File Format

The following table presents the format which the raw data obtained from the Hammond axial jet must be configured it before input into the MATLAB code in A.2. It must be saved as a text file.

#### A.1.1 File Format

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Trans. 1 Zero [V]</td>
<td>Trans. 2 Zero [V]</td>
<td>Trans. 3 Zero [V]</td>
<td>Trans. 4 Zero [V]</td>
<td>Trans. 5 Zero [V]</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Alpha (Pt. 1) [deg]</td>
<td>Beta (Pt. 1) [deg]</td>
<td>Trans. 1 Measured (Pt. 1) [V]</td>
<td>Trans. 2 Measured (Pt. 1) [V]</td>
<td>Trans. 3 Measured (Pt. 1) [V]</td>
<td>Trans. 4 Measured (Pt. 1) [V]</td>
<td>Trans. 5 Measured (Pt. 1) [V]</td>
<td></td>
</tr>
<tr>
<td>Alpha (Pt. 2) [deg]</td>
<td>Beta (Pt. 2) [deg]</td>
<td>Trans. 1 Measured (Pt. 2) [V]</td>
<td>Trans. 2 Measured (Pt. 2) [V]</td>
<td>Trans. 3 Measured (Pt. 2) [V]</td>
<td>Trans. 4 Measured (Pt. 2) [V]</td>
<td>Trans. 5 Measured (Pt. 2) [V]</td>
<td></td>
</tr>
<tr>
<td>.</td>
<td>.</td>
<td>.</td>
<td>.</td>
<td>.</td>
<td>.</td>
<td>.</td>
<td>.</td>
</tr>
</tbody>
</table>
A.1.2 Example File

<table>
<thead>
<tr>
<th>17.039</th>
<th>17.085</th>
<th>17.043</th>
<th>17.094</th>
<th>17.061</th>
<th>99243.66</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000478</td>
<td>0.02943</td>
<td>0.034142</td>
<td>0.007859</td>
<td>0.015428</td>
<td>99360.49</td>
<td>99243.66</td>
</tr>
<tr>
<td>-30</td>
<td>-30</td>
<td>1.220381</td>
<td>-2.999735</td>
<td>2.158437</td>
<td>3.371492</td>
<td>-3.112210</td>
</tr>
<tr>
<td>-20</td>
<td>-30</td>
<td>2.255391</td>
<td>-2.330745</td>
<td>3.626500</td>
<td>2.376296</td>
<td>-2.185910</td>
</tr>
</tbody>
</table>

A.2 MATLAB Code

Once the raw calibration data has been configured into the format presented in A.1, the following MATLAB code can be run in order to obtain coefficients for the probe yaw, probe pitch, total pressure, and static pressure. It will prompt for the name of the text file containing the raw calibration data.

```matlab
% The following code determines Cpp, Cpy, Cpt and Cps for a five hole % pitot probe for use in the raw data reduction m=file. The only % required file for input is a text file of voltages recorded during % the calibration process

% Author: Mike Remaly
% The Pennsylvania State University - University Park, Pa

% Input files:
% 1) user is prompted for raw probe calibration file

% Nomenclature
% cpp - pitch coefficient
% cpy - yaw coefficient
% cpt - total pressure coefficient
% cps - static pressure coefficient
% Pref - ambient pressure during data logging [Pa]
% Ci - calibration constant of transducer 'i' [Pa/V]
% Zi - zero voltage offset of channel 'i' [V]
% Pi - pressure sensed by port 'i' [Pa]
% Pt - total pressure sensed at probe tip [Pa]
% Ps - static pressure sensed at probe tip [Pa]
```
% alpha - pitch angle [deg]
% beta - yaw angle [deg]

clc;
clear all;

disp('|%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%|

disp('| This code generates five-hole probe calibration coefficients|

disp('| using voltages recorded during the calibration process. |')

disp('| Author: Mike Remaly |')

disp('|%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%|

disp('|')

disp('|')

filename=input('Enter raw calibration data file extension: ','s');
RawCalData=[load(filename)];%Open pitot probe calibration data

NCP=length(RawCalData(:,1));

%User inputs - transducer reference pressure and calibration constants
Pref= RawCalData(1,6);
Pt= RawCalData(2,6);
Ps= RawCalData(2,7);

C1= RawCalData(1,1); %Ch1 transducer calibration constant
C2= RawCalData(1,2); %Ch2 transducer calibration constant
C3= RawCalData(1,3); %Ch3 transducer calibration constant
C4= RawCalData(1,4); %Ch4 transducer calibration constant
C5= RawCalData(1,5); %Ch5 transducer calibration constant

Z1= RawCalData(2,1); %Zero offset of channel 1
Z2= RawCalData(2,2); %Zero offset of channel 2
Z3= RawCalData(2,3); %Zero offset of channel 3
Z4= RawCalData(2,4); %Zero offset of channel 4
Z5= RawCalData(2,5); %Zero offset of channel 5

for n=3:NCP;
    alpha= RawCalData(n,1); % read pitch angle
    beta= RawCalData(n,2); % read yaw angle
    P1= (RawCalData(n,3)-Z1)*C1+Pref; %Pressure sensed by port 1
    P2= (RawCalData(n,4)-Z2)*C2+Pref; %Pressure sensed by port 2
    P3= (RawCalData(n,5)-Z3)*C3+Pref; %Pressure sensed by port 3
    P4= (RawCalData(n,6)-Z4)*C4+Pref; %Pressure sensed by port 4
    P5= (RawCalData(n,7)-Z5)*C5+Pref; %Pressure sensed by port 5

    Pbar= .25*(P2+P3+P4+P5); %Average pressure of four outer ports
    Denom= P1-Pbar; %Denominator of coefficient definitions
Cpy= (P2-P3)/Denom;
Cpp= (P4-P5)/Denom;
Cpt= (P1-Pt)/Denom;
Cps= (Pbar-Ps)/Denom;

CalResults(n-2,:)=[alpha beta Cpp Cpy Cpt Cps P1 P2 P3 P4 P5]
end

filename= input('Enter file name for data results: ','s');
dlmwrite(filename,CalResults,'	');

A.3 Resulting Calibration File

The following is the calibration file output by the MATLAB code and modified for use in
the code presented in Appendix B.2. The first column is the probe’s pitch angle relative to the
flow. The second column is the probe’s yaw angle relative to the flow. The third column is the
pitch coefficient, $C_{PP}$. The fourth column is the yaw coefficient, $C_{PY}$. The fifth column is the
total pressure coefficient, $C_{PT}$, and the sixth column is the static pressure coefficient, $C_{PS}$.

The MATLAB code, as written above, does not output in the correct format. When the
yaw angle changes, the pitch angle must always begin at the minimum angle tested, -30° in this
case. For this specific calibration, all the values at a yaw angle of -20°, 0°, and 20° had to be
inverted so that the pitch angle would start at -30°.

<table>
<thead>
<tr>
<th>Pitch Angle</th>
<th>Yaw Angle</th>
<th>Cpp</th>
<th>Cpy</th>
<th>Cpt</th>
<th>Cps</th>
</tr>
</thead>
<tbody>
<tr>
<td>-30</td>
<td>-30</td>
<td>-3.6006</td>
<td>-3.1109</td>
<td>-2.1396</td>
<td>-0.2008</td>
</tr>
<tr>
<td>-20</td>
<td>-30</td>
<td>-1.7895</td>
<td>-2.7408</td>
<td>-1.162</td>
<td>0.10077</td>
</tr>
<tr>
<td>-10</td>
<td>-30</td>
<td>-0.70126</td>
<td>-2.5925</td>
<td>-0.74351</td>
<td>0.17218</td>
</tr>
<tr>
<td>0</td>
<td>-30</td>
<td>0.19246</td>
<td>-2.5626</td>
<td>-0.64631</td>
<td>0.21723</td>
</tr>
<tr>
<td>10</td>
<td>-30</td>
<td>1.1054</td>
<td>-2.6894</td>
<td>-0.79009</td>
<td>0.22926</td>
</tr>
<tr>
<td>20</td>
<td>-30</td>
<td>2.3942</td>
<td>-3.0391</td>
<td>-1.2647</td>
<td>0.1696</td>
</tr>
<tr>
<td>30</td>
<td>-30</td>
<td>4.7595</td>
<td>-3.7486</td>
<td>-2.5567</td>
<td>-0.1252</td>
</tr>
<tr>
<td>-30</td>
<td>-20</td>
<td>-2.8932</td>
<td>-1.6822</td>
<td>-1.1784</td>
<td>0.11016</td>
</tr>
<tr>
<td>-20</td>
<td>-20</td>
<td>-1.5188</td>
<td>-1.5226</td>
<td>-0.52962</td>
<td>0.34413</td>
</tr>
<tr>
<td>-10</td>
<td>-20</td>
<td>-0.58159</td>
<td>-1.4922</td>
<td>-0.25626</td>
<td>0.41563</td>
</tr>
<tr>
<td>0</td>
<td>-20</td>
<td>0.14411</td>
<td>-1.4773</td>
<td>-0.13757</td>
<td>0.45338</td>
</tr>
<tr>
<td>10</td>
<td>-20</td>
<td>0.88097</td>
<td>-1.4844</td>
<td>-0.24791</td>
<td>0.46111</td>
</tr>
<tr>
<td>20</td>
<td>-20</td>
<td>1.9104</td>
<td>-1.6374</td>
<td>-0.59351</td>
<td>0.39237</td>
</tr>
<tr>
<td>30</td>
<td>-20</td>
<td>3.4503</td>
<td>-1.8892</td>
<td>-1.3589</td>
<td>0.12608</td>
</tr>
<tr>
<td>-----</td>
<td>------</td>
<td>--------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
</tr>
<tr>
<td>-30</td>
<td>-10</td>
<td>-2.4757</td>
<td>-0.79043</td>
<td>-0.69651</td>
<td>0.20535</td>
</tr>
<tr>
<td>-20</td>
<td>-10</td>
<td>-1.3859</td>
<td>-0.67623</td>
<td>-0.22932</td>
<td>0.43664</td>
</tr>
<tr>
<td>-10</td>
<td>-10</td>
<td>-0.55343</td>
<td>-0.65625</td>
<td>-0.03777</td>
<td>0.52137</td>
</tr>
<tr>
<td>0</td>
<td>-10</td>
<td>0.11721</td>
<td>-0.65436</td>
<td>0.005549</td>
<td>0.53135</td>
</tr>
<tr>
<td>10</td>
<td>-10</td>
<td>0.81998</td>
<td>-0.6102</td>
<td>-0.0391</td>
<td>0.5456</td>
</tr>
<tr>
<td>20</td>
<td>-10</td>
<td>1.6967</td>
<td>-0.67029</td>
<td>-0.27397</td>
<td>0.45283</td>
</tr>
<tr>
<td>30</td>
<td>-10</td>
<td>2.8348</td>
<td>-0.72317</td>
<td>-0.7956</td>
<td>0.22222</td>
</tr>
<tr>
<td>-30</td>
<td>0</td>
<td>-2.312</td>
<td>-0.07194</td>
<td>-0.57319</td>
<td>0.22351</td>
</tr>
<tr>
<td>-20</td>
<td>0</td>
<td>-1.3531</td>
<td>-0.01165</td>
<td>-0.15409</td>
<td>0.44966</td>
</tr>
<tr>
<td>-10</td>
<td>0</td>
<td>-0.58003</td>
<td>0.017583</td>
<td>-0.00077</td>
<td>0.53232</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0.11785</td>
<td>0.042031</td>
<td>0.028939</td>
<td>0.5376</td>
</tr>
<tr>
<td>10</td>
<td>0</td>
<td>0.83233</td>
<td>0.061259</td>
<td>-0.02603</td>
<td>0.54816</td>
</tr>
<tr>
<td>20</td>
<td>0</td>
<td>1.6609</td>
<td>0.10223</td>
<td>-0.23915</td>
<td>0.46111</td>
</tr>
<tr>
<td>30</td>
<td>0</td>
<td>2.7943</td>
<td>0.17148</td>
<td>-0.72101</td>
<td>0.21593</td>
</tr>
<tr>
<td>-30</td>
<td>10</td>
<td>-2.4175</td>
<td>0.67481</td>
<td>-0.62592</td>
<td>0.22813</td>
</tr>
<tr>
<td>-20</td>
<td>10</td>
<td>-1.423</td>
<td>0.64919</td>
<td>-0.21854</td>
<td>0.45046</td>
</tr>
<tr>
<td>-10</td>
<td>10</td>
<td>-0.59041</td>
<td>0.67164</td>
<td>-0.022</td>
<td>0.53626</td>
</tr>
<tr>
<td>0</td>
<td>10</td>
<td>0.1026</td>
<td>0.72253</td>
<td>-0.01011</td>
<td>0.54122</td>
</tr>
<tr>
<td>10</td>
<td>10</td>
<td>0.81699</td>
<td>0.74807</td>
<td>-0.06992</td>
<td>0.53758</td>
</tr>
<tr>
<td>20</td>
<td>10</td>
<td>1.7098</td>
<td>0.84477</td>
<td>-0.31156</td>
<td>0.44712</td>
</tr>
<tr>
<td>30</td>
<td>10</td>
<td>2.9939</td>
<td>1.0392</td>
<td>-0.94193</td>
<td>0.196</td>
</tr>
<tr>
<td>-30</td>
<td>20</td>
<td>-2.9247</td>
<td>1.621</td>
<td>1.0734</td>
<td>0.16062</td>
</tr>
<tr>
<td>-20</td>
<td>20</td>
<td>-1.6346</td>
<td>1.5246</td>
<td>-0.51157</td>
<td>0.39519</td>
</tr>
<tr>
<td>-10</td>
<td>20</td>
<td>-0.67186</td>
<td>1.5028</td>
<td>-0.27076</td>
<td>0.45392</td>
</tr>
<tr>
<td>0</td>
<td>20</td>
<td>0.090247</td>
<td>1.5319</td>
<td>-0.1941</td>
<td>0.47106</td>
</tr>
<tr>
<td>10</td>
<td>20</td>
<td>0.86306</td>
<td>1.6279</td>
<td>-0.30367</td>
<td>0.45143</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>1.9123</td>
<td>1.8427</td>
<td>-0.64379</td>
<td>0.3756</td>
</tr>
<tr>
<td>30</td>
<td>20</td>
<td>3.5814</td>
<td>2.2544</td>
<td>-1.468</td>
<td>0.12885</td>
</tr>
<tr>
<td>-30</td>
<td>30</td>
<td>-3.8459</td>
<td>3.1203</td>
<td>-2.0935</td>
<td>-0.1204</td>
</tr>
<tr>
<td>-20</td>
<td>30</td>
<td>-2.0153</td>
<td>2.8094</td>
<td>-1.184</td>
<td>0.18331</td>
</tr>
<tr>
<td>-10</td>
<td>30</td>
<td>-0.83739</td>
<td>2.6415</td>
<td>-0.74231</td>
<td>0.2332</td>
</tr>
<tr>
<td>0</td>
<td>30</td>
<td>0.086268</td>
<td>2.6502</td>
<td>-0.66637</td>
<td>0.25196</td>
</tr>
<tr>
<td>10</td>
<td>30</td>
<td>1.0404</td>
<td>2.8776</td>
<td>-0.85967</td>
<td>0.21472</td>
</tr>
<tr>
<td>20</td>
<td>30</td>
<td>2.3784</td>
<td>3.3645</td>
<td>-1.3879</td>
<td>0.15672</td>
</tr>
<tr>
<td>30</td>
<td>30</td>
<td>4.8452</td>
<td>4.3977</td>
<td>-2.7829</td>
<td>-0.2015</td>
</tr>
</tbody>
</table>
APPENDIX B

MATLAB CODE AND FILE FORMATS FOR PROCESSING FIVE-HOLE PITOT PROBE DATA FROM AN UNKNOWN FLOW

B.1 FILE FORMAT FOR RAW DATA OF UNKNOWN FLOW

The raw data obtained from one point within the unknown flow must be input into this format before it can be used in the code in B.2.

B.1.1 File Format

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Trans. 1 Zero [V]</td>
<td>Trans. 2 Zero [V]</td>
<td>Trans. 3 Zero [V]</td>
<td>Trans. 4 Zero [V]</td>
<td>Trans. 5 Zero [V]</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>Trans. 1 Measured [V]</td>
<td>Trans. 2 Measured [V]</td>
<td>Trans. 3 Measured [V]</td>
<td>Trans. 4 Measured [V]</td>
<td>Trans. 5 Measured [V]</td>
</tr>
</tbody>
</table>

B.1.2 Example File

```
23.77778 99243.66 17.039 17.085 17.043 17.094 17.061
0.000478 0.02943 0.034142 0.007859 0.015428 0 0
0 0 -0.16398 -0.87237 -0.88075 -0.75389 -0.95432
```
B.2 MATLAB Code

The following MATLAB code will prompt for three required file names before running. The first is the name of the calibration file presented in A.3. The second is the raw data obtained from the unknown flow, B.1, and the third is the name of the file which the code will output the results to, B.3.

```matlab
% The following routine calculates the mean velocity components from
% five-hole pitot probe data. The results include the flow velocity and
% direction
% Author: Mike Remaly and Dr. Camci
% The Pennsylvania State University - University Park, PA
% Edited by Jason Halwick - July 21 2010
% Change Log
% Line 107: commented out '*100', data is already in Pa
% Lines 166-176: added warning if alpha is out of cal. range
% Lines 217-227: added warning if beta is out of cal. range

% Input files:
% 1) user is prompted for probe calibration file
% 2) user is prompted for raw five-hole probe data

% Nomenclature
%______________________________________________________________________
% cpp - pitch coefficient from probe calibration
% cpy - yaw coefficient from probe calibration
% cpt - total pressure coefficient from probe calibration
% To - ambient temperature during data logging [K]
% Pref - ambient pressure during data logging [Pa]
% Ci - calibration constant of transducer 'i'[Pa/V]
% Zi - zero voltage offset of channel 'i'[V]
% x - x-coordinate of data point
% y - y-coordinate of data point
% pi - pressure sensed by port 'i'[Pa]
% cpyts - experimental yaw coefficent
% cppts - experimental pitch coefficient
% alpha - pitch angle[deg]
% beta - yaw angle [deg]
% pt - total pressure sensed at probe tip [Pa]
% ps - static pressure sensed at probe tip [Pa]
% Vmag - velocity magnitude sensed by probe [m/s]
%______________________________________________________________________

% Matlab Command Screen Formatting and Inout File Prompts
%______________________________________________________________________
clear all;
format short;
```
disp('|%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%|
| This code reduces data measured by a five hole pitot probe. |
| The output file contains the flow direction and magnitude. |
| Author: Mike Remaly & Dr. Camci |
|%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%|

filename= input('Enter calibration data file with extension: ','s');
CalibrationData= [load(filename)]; %Open fhp pitot probe cal. data

filename= input('Enter file with extension: ','s');
TestData= [load(filename)]; %Open raw test data

% Calibration parameters
%______________________________________________________________________
AbsAngleMax= 30; %Max calibration angle [deg]
AngleIncrement=10;
NCP= (2*AbsAngleMax/AngleIncrement)+1; %Number of calibration pts for each beta
Results= zeros(NCP,10); %create empty matrix for results

AlphaVec= (-1*AbsAngleMax:AngleIncrement:AbsAngleMax)'; %Pitch angles used for calibration
BetaVec= (-1*AbsAngleMax:AngleIncrement:AbsAngleMax)'; %Yaw angles used for calibration

% Read in calibration data and plot Cpp, Cpy, Cpt, Cps
%______________________________________________________________________
n=0;
for j=1:NCP
    cpp(:,j)= CalibrationData(((n*NCP)+1):((n*NCP)+NCP),3); %Generate Cpp matrix
    cpy(:,j)= CalibrationData(((n*NCP)+1):((n*NCP)+NCP),4); %Generate Cpy matrix
    cpt(:,j)= CalibrationData(((n*NCP+1)):((n*NCP+NCP),5); %Generate Cpt matrix
    cps(:,j)= CalibrationData(((n*NCP+1)):((n*NCP+NCP),6); %Generate Cps matrix
    n=n+1;
end

figure(1)
plot(cpp,cpy,'ks','MarkerFaceColor','k') % Plot Cpy vs. Cpp
xlabel('Cpp')
ylabel('Cpy')
axis([-6 6 -5 5])
grid on
set(gcf, 'color','w')
for j=1:NCP
    AlphaPlot(:,j)=AlphaVec; %Generate matrix of pitch angles
end

% figure(2)
% plot(AlphaVec,cps,'k') %Plot of Cps vs. Alpha
% ylabel('Static Pressure Coefficient')
% xlabel('Pitch Angle [deg]')
% axis([-35 35 -.4 .8])
% grid on
% set(gcf,'color','w')
% figure(3)
% plot(AlphaVec,cpt,'k') %Plot of Cpt vs. Alpha
% ylabel('Total pressure Coefficient')
% xlabel('Pitch Angle [deg]')
% axis([-35 35 -3.5 1])
% grid on
% set(gcf,'color','w')

% Calculate Air density at Time of Data Acquision
%______________________________________________________________________
To=TestData(1,1)+273; %Convert from Celsius to Kelvin
Pref= TestData(1,2);%*100; %Convert mbar to Pa % '*100' omitted, data in Pa
rho= Pref/(287*To); %Calculate air density
% Read tranducer calibration constants and zero voltage offsets
%______________________________________________________________________
C1=TestData(1,3); %Ch1 transducer calibration constant
C2=TestData(1,4); %Ch2 transducer calibration constant
C3=TestData(1,5); %Ch3 transducer calibration constant
C4=TestData(1,6); %Ch4 transducer calibration constant
C5=TestData(1,7); %Ch5 transducer calibration constant
Z1= TestData(2,1); % Zero offset of channel 1
Z2= TestData(2,2); % Zero offset of channel 2
Z3=TestData(2,3); % Zero offset of channel 3
Z4= TestData(2,4); % Zero offset of channel 4
Z5= TestData(2,5); % Zero offset of channel 5
% Begin Main Loop of Velocity Component Calculation
%______________________________________________________________________
for k=3:length(TestData(:,1)) %Read line by line the raw experimental data
    x=TestData(k,1);
    y=TestData(k,2);
    p1=C1*(TestData(k,3)-Z1); %Pressure sensed by port 1
    p2=C2*(TestData(k,4)-Z2); %Pressure sensed by port 2
    p3=C3*(TestData(k,5)-Z3); %Pressure sensed by port 3
    p4=C4*(TestData(k,6)-Z4); %Pressure sensed by port 4
    p5=C5*(TestData(k,7)-Z5); %Pressure sensed by port 5

pbar = 0.25*(p2+p3+p4+p5);  \% Average pressure of 4 outer ports  
D = p1-pbar;  \% Denominator of non-dimensional coefficient  

cpyts = (p2-p3)/D;  
cppts = (p4-p5)/D;  

\% Find Alpha ___________________  
for i=1:NCP  
  for j=2:(NCP-1);  
    if(cpy(i,j)>=cpyts)  
      break  
    elseif(j==NCP-1)  
      j=NCP;  
    end  
  end  

\% Linear Interpolation to get Cpp for each alpha  
f(i)=(cpp(i,j)-cpp(i,j-1))/(cpy(i,j)-cpy(i,j-1))*(cpyts-cpy(i,j-1))+cpp(i,j-1);  

end  
for i=2:(NCP-1);  
  if(f(i)>=cppts)  
    break  
  elseif(i==(NCP-1))  
    i=NCP;  
  end  
end  

\% Linear Interpolation to solve for alpha  
alpha=(AlphaVec(i)-AlphaVec(i-1))/(f(i)-f(i-1))*(cppts-f(i-1))+AlphaVec(i-1);  

\%  Added by Jason  
if(abs(alpha)>35)  
  disp("|---------------------------------------------------|");  
  disp("|");  
  disp("| Warning: Alpha is out of calibration range |");  
  disp("|");  
  disp("|---------------------------------------------------|");  
end  
\% end Addition  
\%  
if(abs(alpha)>35)  \% Execute statement if out of calibration range  
  Results(k,1:2)=[x y];  
  Results(k,3:4)=0;  
end
if(abs(alpha)<=35) % Continue if alpha is in calibration range
for i=2:(NCP-1)
    if(AlphaVec(i)>=alpha)
        break
    elseif(i==(NCP-1))
        i=NCP;
    end
end
ip=i;
im=i-1;

for j=1:NCP %Linear Interpolation to get Cpp for each alpha
    f(j)=(cpy(ip,j)-cpy(im,j))/(cpp(ip,j)-cpp(im,j))*(cppts-cpp(im,j))+cpy(im,j);
end

for j=2:(NCP-1)
    if(f(j)>=cpyts)
        break
    elseif(j==(NCP-1))
        j=NCP;
    end
end
jp=j;
jm=j-1;

% Linear Interpolation to solve for beta
beta= (BetaVec(jp)-BetaVec(jm))/(f(jp)-f(jm))*(cpyts-f(jm))+BetaVec(jm);
end

if(abs(beta)>35) % Execute statement if out of calibration range
    Results(k,1:2)=x y;
    Results(k,3:4)=0;
end

if(abs(beta)<=35) % Proceed if beta is in calibration range
% Find Cp_total and Cp_static

% Linear Interpolation to find lower limit for lower beta limit to get Cpt
\[ cptp = \frac{(cpt(ip,jp)-cpt(im,jp))}{(AlphaVec(ip)-AlphaVec(im))} \alpha - AlphaVec(im) + cpt(im,jp) \]

% Linear Interpolation
\[ cptm = \frac{(cpt(ip,jm)-cpt(im,jm))}{(AlphaVec(ip)-AlphaVec(im))} \alpha - AlphaVec(im) + cpt(im,jm) \]

\[ cptt = \frac{(cptp-cptm)}{(BetaVec(jp)-BetaVec(jm))} (beta - BetaVec(jm)) + cptm \]

% Linear interpolation to find lower alpha limit for lower beta limit to get Cpt
\[ cpsp = \frac{(cps(ip,jp)-cps(im,jp))}{(AlphaVec(ip)-AlphaVec(im))} \alpha - AlphaVec(im) + cps(im,jp) \]

% Linear Interpolation
\[ cpsm = \frac{(cps(ip,jm)-cps(im,jm))}{(AlphaVec(ip)-AlphaVec(im))} \alpha - AlphaVec(im) + cps(im,jm) \]

\[ cpss = \frac{(cpsp-cpsm)}{(BetaVec(jp)-BetaVec(jm))} (beta - BetaVec(jm)) + cpsm \]

% Total and Static Pressure Calculation
\[ pt = p1 - cptt*D \]
\[ ps = pbar - cpss*D \]
\[ pt = pt + Pref \]
\[ ps = ps + Pref \]

% Velocity Manitude Calculation
\[ \text{if}(ps > pt) \]
\[ \text{ps} = pt; \]
\[ \text{end} \]
\[ Vmag = ((2*(pt-ps))/\rho)^{.5}; \]

% Velocity Components Calculation
\[ calpha = \cos(\alpha*\pi/180); \]
\[ cbeta = \cos(\beta*\pi/180); \]
\[ salpha = \sin(\alpha*\pi/180); \]
\[ sbeta = \sin(\beta*\pi/180); \]

\[ u = Vmag*cbeta*calpha; \]
\[ v = -1*Vmag*sbeta; \]
\[ w = -1*Vmag*salpha*cbeta; \]

% Store all results in single matrix
Results(k,:) = [x y pt ps u v w Vmag alpha beta]; %write output

end

end % end of main loop
% Write results to file
%______________________________________________________________________
filename=input('Enter file name for data results: ','s');
dlmwrite(filename,Results,'\t')

B.3 FILE FORMAT OF OUTPUT

Below is how the MATLAB code in B.2 will output the results of the unknown flow sensed by the five-hole pitot probe when examining one location at a time. The ‘x’ and ‘y’ refer to probe location and will be zero for this use. $P_T$ is the total pressure of the flow. $P_s$ is the static pressure of the flow. Both are presented in Pascals. The $u$, $v$, and $w$ are velocity components of the flow in m/sec with respect to the probe as described in Figure 3-2. $V_{MAG}$ is the velocity magnitude in m/sec. Alpha is the pitch angle, and beta is the yaw angle, both in degrees.

```
0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0
x  y  $P_T$  $P_s$  $u$  $v$  $w$  $V_{MAG}$  alpha  beta
0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0
```

APPENDIX C

MEASUREMENTS OF SELECT SHAPES OF THE DUAL-DUCTED FAN MODEL

The following figures display the comparison of the CAD file used for the CFD fuselage mesh (red solid body) to measurements obtained off the physical model using a dial indicator.

Figure C-1: Locations of Select Shape Measurements

Front Face
• Cross-section of the front of forward duct

• Measured points shown circled in blue

• Black line shows approximated cross-section

• Maximum estimated difference is 0.115 inches

Figure C-2: Measurements of Front Face of the Forward Duct
**Forward Duct - Side**

- Cross-section of the side of forward duct
- Measured Points shown circled in blue
- Black line shows approximated cross-section
- Largest estimated difference is 0.189 inches

**Figure C-3: Measurements of Side of the Forward Duct**

**Center of Fuselage**

- Cross-section of the fuselage center, along center line
- Measured Points shown circled in blue
- Black line shows approximated cross-section
- Largest estimated difference is 0.252 inches along bottom-aft corner
- Estimated difference at top-aft corner is 0.156 inches
- Estimated difference at top-forward corner is 0.092 inches

**Figure C-4: Measurements of Centerline Cut of the Center-body**
• Cross-section of the fuselage center, offset 1.75 inches from the centerline

• Measured Points shown circled in blue

• Black line shows approximated cross-section

• Estimated difference at bottom-aft corner is 0.282 inches

• Estimated difference at bottom-forward corner is 0.404 inches

Figure C-5: Measurements of Offset Cut (1.75 inches) of Center-body