CAVITATION ASSESSMENT AND HYDRODYNAMIC
PERFORMANCE CHARACTERISTICS OF A MARINE
HYDROKINETIC TURBINE FOIL DESIGN

A Thesis in
Aerospace Engineering
by
Ryan S. Phillips

© 2012 Ryan S. Phillips

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Master of Science

August 2012
The thesis of Ryan S. Phillips was reviewed and approved* by the following:

Dennis K. McLaughlin  
Professor of Aerospace Engineering, Thesis Co-Advisor  
Aerospace Engineering Department Advisor

Arnold A. Fontaine  
Senior Scientist, Head of Fluid Dynamics Division  
Applied Research Laboratory  
Thesis Co-Advisor

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

William A. Straka  
R & D Engineer, Applied Research Laboratory

*Signatures are on file in the Graduate School.
Abstract

A uniquely designed hydrofoil for use on MHK turbines was tested in a water tunnel at The Pennsylvania State University’s Applied Research Lab Garfield Thomas Water Tunnel. Baseline tests were conducted with a NACA 4412 foil and compared with historical data to validate the facility and test procedures. Lift and drag data were acquired using a six degree of freedom load cell. Drag data calculated from LDV wake profiles showed good agreement with the load cell data. The new hydrofoil, MHKF1-180s, was designed for improved cavitation performance with lower sensitivity to roughness and reduced probability to trailing edge singing. The MHKF1-180s exhibited improved cavitation performance when compared to the NACA 4412. The sensitivity to both distributed and isolated roughness was studied on the MHKF1-180s hydrofoil with the overall conclusion that isolated roughness affected the cavitation performance more dramatically than distributed roughness and the effect was greater with larger particle size. The NACA 4412 also had a lower sensitivity to roughness than the MHKF1-180s but this is likely due to a thickness effect. This result proved that foiling or damage sensitivity is an important design consideration. The experimental water tunnel data was also compared to Reynolds averaged Navier Stokes CFD simulations and there was variation in the results likely due to a transitional effect. An LDV spectral analysis was performed on the trailing edge to capture vortex shedding and the conclusion was that the frequency spikes in the spectra data were related to the reference height of the trailing edge singing geometry.
# Table of Contents

List of Figures vi
List of Tables xi
List of Symbols xii
Acknowledgments xvi

Chapter 1

Introduction 1
1.1 Objectives ................................................. 1
1.2 Motivation ................................................... 2
1.3 Siting Options for MHK Technologies ......................... 4
1.4 Environmental Impacts ........................................ 11
  1.4.1 Impact on Aquatic Life and Local Habitat ............... 12
1.5 Existing Marine Hydrokinetic Devices and Designs .......... 13
  1.5.1 Support Methods ....................................... 14
  1.5.2 Horizontal Axis Devices ................................ 19
1.6 Blade and Foil Design ....................................... 36
  1.6.1 Blade Element Momentum Theory (BEMT) and the Betz Limit 36
  1.6.2 Design Criteria ......................................... 39
  1.6.3 Trailing Edge Singing ................................... 43
  1.6.4 Roughness Sensitivity ................................... 45
  1.6.5 Cavitation .............................................. 48
1.7 NACA 4412 Historical Data .................................... 53

Chapter 2

Experimental Setup 59
2.1 Applied Research Lab 12-Inch Diameter Water Tunnel Facility ......................................................... 59
Chapter 3

Data Reduction and Corrections

3.1 Data Reduction Equations

3.2 Gap Corrections Applied to Load Data

3.3 Tunnel Blockage Corrections

3.4 Error Analysis

3.5 Reynolds Number Effects

3.6 LDV Wake Profiles

Chapter 4

Test Results

4.1 Test Plan

4.2 Uniform Inflow Study

4.3 NACA 4412 Test Validation

4.3.1 Force Results

4.3.2 Cavitation Results

4.3.3 Flow Visualization

4.4 MHKF1-180 Hydrofoil Design Test Results

4.4.1 Force Results

4.4.2 Cavitation Results

4.4.3 Flow Visualization

4.5 Trailing edge Spectra Analysis

4.6 CFD Comparison

Chapter 5

Conclusions and Future Work

5.1 Summary and Conclusions

5.2 Future Work

References
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Cross section of foil designed tested in this thesis for the Sandia marine hydrokinetic turbine.</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>Diagram of spring and neap tides.</td>
<td>5</td>
</tr>
<tr>
<td>1.3</td>
<td>Tidal map showing the amplitude of the tide. The contours are divided into 10 cm segments.</td>
<td>6</td>
</tr>
<tr>
<td>1.4</td>
<td>Pentland Firth area map.</td>
<td>7</td>
</tr>
<tr>
<td>1.5</td>
<td>Bay of Fundy tidal resource map assuming that 15% of the tidal energy can be extracted without significant environmental harm.</td>
<td>8</td>
</tr>
<tr>
<td>1.6</td>
<td>Mississippi River hydrokinetic project map.</td>
<td>9</td>
</tr>
<tr>
<td>1.7</td>
<td>Image of the Gulf Stream including the Florida Current. Black dots refer to velocities greater than 1 m/s.</td>
<td>10</td>
</tr>
<tr>
<td>1.8</td>
<td>Example of a monopile mounted MHK turbine</td>
<td>14</td>
</tr>
<tr>
<td>1.9</td>
<td>Example of a gravity or anchored support structure</td>
<td>16</td>
</tr>
<tr>
<td>1.10</td>
<td>Example of an anchored MHK turbine with buoyant nacelle</td>
<td>16</td>
</tr>
<tr>
<td>1.11</td>
<td>Example of a floating moored MHK turbine</td>
<td>17</td>
</tr>
<tr>
<td>1.12</td>
<td>Example of a sheath system MHK turbine</td>
<td>18</td>
</tr>
<tr>
<td>1.13</td>
<td>Horizontal open rotor hydrokinetic devices</td>
<td>22</td>
</tr>
<tr>
<td>1.14</td>
<td>MHK devices with turbines aligned on the same axis</td>
<td>26</td>
</tr>
<tr>
<td>1.15</td>
<td>Examples of multiple turbines attacked to the same support structure</td>
<td>27</td>
</tr>
<tr>
<td>1.16</td>
<td>Pile and gravity moored MHK turbines</td>
<td>27</td>
</tr>
<tr>
<td>1.17</td>
<td>Positively buoyant MHK turbines tethered to the seafloor</td>
<td>28</td>
</tr>
<tr>
<td>1.18</td>
<td>Diagram of a diffuser with station labels</td>
<td>31</td>
</tr>
<tr>
<td>1.19</td>
<td>Power augmentation caused by a duct. The blue dotted line corresponds to the exit pressure coefficient of a horizontal axis turbine without a duct.</td>
<td>33</td>
</tr>
<tr>
<td>1.20</td>
<td>Symmetric duct hydrokinetic devices.</td>
<td>35</td>
</tr>
<tr>
<td>1.21</td>
<td>Open Hydro Open Centre turbine.</td>
<td>35</td>
</tr>
<tr>
<td>1.22</td>
<td>Augmented turbine designs for uni-directional flow.</td>
<td>36</td>
</tr>
<tr>
<td>1.23</td>
<td>Diagram showing the stream tube pressure drop and velocities at several stations.</td>
<td>37</td>
</tr>
<tr>
<td>1.24</td>
<td>Blade section showing the velocity triangle.</td>
<td>38</td>
</tr>
<tr>
<td>1.25</td>
<td>The effect of blade number on power coefficient of an optimum rotor with zero drag.</td>
<td>41</td>
</tr>
</tbody>
</table>
1.26 The effect of solidity on power coefficient. .................................. 41
1.27 Lift to drag ratio effect on power for a three bladed optimum rotor. . . 42
1.28 MHKF1 family of foils. ............................................................. 43
1.29 MHKF1-180s (blue) trailing edge modified to the MHKF1-180c (red) by removing the hashed area. ........................................... 45
1.30 Non-dimensional roughness height as a function of the ratio of roughness Reynolds number and station Reynolds number. .................. 48
1.31 The effect of gas content on cavitation inception on a 1.5 caliber ogive. . 51
1.32 Figures from Shen and Dimotakis showing the cavitation effect on hydrodynamic forces. .......................................................... 54
1.33 NACA 4412 historical lift data at comparable Reynolds numbers. ....... 55
1.34 NACA 4412 historical drag data at comparable Reynolds numbers. .... 55
1.35 Schematic of the NACA variable density wind tunnel. .................... 57
1.36 Schematic of the NACA low turbulence wind tunnel. ....................... 57
1.37 The dependency of turbulence intensity on model chord for the NACA low turbulence tunnel. ............................................... 58
1.38 The Texas A&M Wind Tunnel. ................................................... 58
2.1 PSU/ARL 12-inch diameter water tunnel schematic illustrating both circular and rectangular test sections. ............................... 60
2.2 PSU/ARL 12-inch diameter water tunnel window modifications. .......... 61
2.3 Solid foil and mounting post two-dimensional cross sections. ............. 62
2.4 NACA 4412 solid fin installed in the test section showing gaps. ........... 63
2.5 NACA 4412 three part fin CAD drawing. ....................................... 63
2.6 Exploded view of the 3 part test assembly. .................................... 65
2.7 LDV hardware setup. ................................................................. 68
2.8 Flow visualization techniques. .................................................... 69
2.9 Photographs close to the point of calling cavitation. ......................... 71
2.10 Locations of isolated roughness elements based on XFOIL analysis of minimum pressure location. ........................................ 72
2.11 Methods of boundary layer tripping. .......................................... 73
2.12 LDV trailing edge survey grid used to find the location of maximum RMS. Red dots indicate locations velocity was measured. .......... 73
3.1 Coordinate system for tunnel tests. ............................................. 75
3.2 Gap effect on the force coefficients for two angles of attack. .............. 77
3.3 Change in $c_l$ for different gap sizes as a function of angle of attack. .. 78
3.4 Change in $c_d$ for different gap sizes as a function of angle of attack. .. 79
3.5 Lambda 2 Factor for different streamline shapes. Fineness ratio refers to chord/thickness ................................................... 80
3.6 Lift and L/D curves for the NACA 4412 over a large Reynolds number range. ................................................................. 84
3.7 Influence of Reynolds number on surface roughness effects. .............. 84
3.8 Reynolds number effect on cavitation number on the MHKF1-180s foil with isolated roughness elements.  
3.9 Wake profile for the MHKF1-180s at -4 degrees angle of attack.  
4.1 Locations where the vertical uniform velocity traverses were taken.  
4.2 Vertical uniform inflow survey taken 2.5 inches upstream of the fin at several locations along the span of the test section.  
4.3 Horizontal Uniform inflow survey at the tunnels centerline.  
4.4 NACA 4412 lift data comparison with historic data.  
4.5 NACA 4412 drag data comparison with historic data.  
4.6 NACA 4412 l/d data comparison with historic data.  
4.7 NACA 4412 desinent cavitation performance compared with historical water tunnel data. Error bars represent the range of observed data calls.  
4.8 NACA 4412 cavitation hysteresis with 60 grit roughness applied to the leading edge.  
4.9 NACA 4412 cavitation hysteresis with 46 grit isolated roughness applied to the foil.  
4.10 NACA 4412 cavitation hysteresis with 16 grit isolated roughness applied to the foil.  
4.11 NACA 4412 lift breakdown in the clean condition.  
4.12 NACA 4412 drag breakdown in the clean condition.  
4.13 NACA 4412 lift breakdown with 60 grit distributed roughness applied to the leading edge.  
4.14 NACA 4412 drag breakdown with 60 grit distributed roughness applied to the leading edge.  
4.15 Oil paint flow visualization at 5 degrees angle of attack and 1.3 million Reynolds number showing two-dimensional flow.  
4.16 Oil paint flow visualization at 15 degrees angle of attack and 1.3 million Reynolds number showing separated flow and loss of two dimensionality.  
4.17 MHKF1-180s, MHKF1-180c, NACA 4412 and NACA 4418 lift compared in the clean condition.  
4.18 MHKF1-180s, MHKF1-180c, NACA 4412 and NACA 4418 drag compared in the clean condition.  
4.19 MHKF1-180s, MHKF1-180c, NACA 4412 and NACA 4418 L/D comparison in the clean condition.  
4.20 MHKF1-180s and NACA 4412 lift compared at several roughness cases at 1.3 million Reynolds number.  
4.21 MHKF1-180s and NACA 4412 drag data compared at several roughness cases at 1.3 million Reynolds number.  
4.22 MHKF1-180s L/D data at 1.3 million Reynolds number.  
4.23 MHKF1-180s and NACA 4412 roughness sensitivity comparison.
4.24 Comparison of the Abbott 6 million Reynolds number data for the NACA 4412 and NACA 4418 in a clean and rough condition. 107
4.25 Foil clean desinent cavitation performance comparison. 108
4.26 Photographs comparing developed cavitation on the NACA 4412 and MHKF1-180s at the same condition. 108
4.27 MHKF1-180s cavitation hysteresis under the clean condition. 109
4.28 MHKF1-180s cavitation hysteresis with 60 grit roughness applied to the leading edge. 109
4.29 MHKF1-180s cavitation hysteresis with 46 grit roughness applied to the leading edge. 110
4.30 MHKF1-180s cavitation hysteresis with 16 grit roughness applied to the leading edge. 110
4.31 NACA 4412 desinent cavitation performance with roughness elements added. 113
4.32 Isolated and distributed roughness effect on desinent cavitation number on the MHKF1-180s foil. 113
4.33 Cavitation roughness sensitivity comparison for the MHKF1-180s and NACA 4412. 114
4.34 Comparison of distributed and isolated roughness on the MHKF1-180s at a similar condition. 114
4.35 MHKF1-180s example of cavity length when breakdown is about to occur. The red lines represent the unsteadiness of the cavity length. 115
4.36 MHKF1-180s cavitation lift breakdown. 116
4.37 MHKF1-180s cavitation drag breakdown. 116
4.38 MHKF1-180s cavitation lift breakdown with distributed roughness. 117
4.39 MHKF1-180s cavitation drag breakdown with distributed roughness. 117
4.40 Paint flow visualization for separated and attached flow at 1.3 million Reynolds number. 118
4.41 Wake survey for all three foils tested. 120
4.42 Theoretical and experimental comparison of shedding frequency based on the trailing edge thickness length scale. 121
4.43 Trailing edge LDV spectra data comparing the three foils at 5 meters per second. 122
4.44 Trailing edge LDV spectra data comparing the three foils at 6 meters per second. 122
4.45 Trailing edge LDV spectra data comparing the three foils at 8 meters per second. 123
4.46 Trailing edge LDV spectra data comparing the three foils at 10 meters per second. 123
4.47 MHKF1-180s tunnel grid at 0 degrees angle of attack. 124
4.48 MHKF1-180s tunnel grid at 0 degrees angle of attack. 125
4.49 NACA 4412 lift comparison of CFD and experimental data. 127
4.50 NACA 4412 drag comparison of CFD and experimental data. . . . . . . 128
4.51 MHKF1-180s lift comparison of CFD and experimental data. . . . . . . 129
4.52 MHKF1-180s drag comparison of CFD and experimental data. . . . . . 129
4.53 OVERFLOW and XFOIL analysis of MHKF1-180s and NACA 4418 foil. 130
List of Tables

1.1 The top 80 percent of all the total tidal resources in the UK . . . . . . . 7
1.2 Environmental stressors and receptors . . . . . . . . . . . . . . . . . . 11
1.3 Single open rotor MHK devices . . . . . . . . . . . . . . . . . . . . . . 21
1.4 Multiple rotor horizontal axis devices . . . . . . . . . . . . . . . . . . 24
1.5 Augmented horizontal axis devices . . . . . . . . . . . . . . . . . . . . 34

2.1 AMTI SP2.5D-1K-6010 load cell specifications . . . . . . . . . . . . . 66
2.2 AMTI SP2.5D-1K-6010 load cell capacities and sensitivities . . . . . . 66
2.3 LDV laser probe volume properties . . . . . . . . . . . . . . . . . . . . 67
2.4 Oil-Paint to gear oil mix ratios based on testing speed . . . . . . . . . 69

3.1 Barlow et. al. blockage correction equations . . . . . . . . . . . . . . . 80
3.2 Load cell errors provided by AMTI . . . . . . . . . . . . . . . . . . . . 81

4.1 Test Matrix . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 88
4.2 Trailing edge survey wake data for each foil tested . . . . . . . . . . . . 119
List of Symbols

Ω  Rotor Speed
α  Angle of Attack
α_u Uncorrected Angle of Attack
β  Blade Twist Angle
γ  Diffuser Velocity Ratio
δ_1 Laminar Sub-Layer Thickness
δ_q Error in a quantity where q is any quantity
ε_{sb} Solid Blockage Factor
ε_{wb} Wake Blockage Factor
η_d Diffuser Efficiency
η_k Non-Dimensional Roughness Height
θ  Blade twist angle
λ  Tip Speed Ratio (TSR)
λ_2 Lambda 2 Factor Approximating an Airfoil Shape from a Cylinder
λ_h Tip Speed Ratio at Turbine Hub
λ_r Local Tip Speed Ratio
μ  Dynamic Viscosity
ν  Kinematic Viscosity
ρ  Fluid Density

xii
\( \sigma \) Cavitation Number

\( \sigma_{\text{Block}} \) Blockage Correction Sigma Factor

\( \tau \) Wall Shear Stress

\( \phi \) Flow Angle

\( \omega \) Local Rotor Speed

\( A \) Area of a Surface

\( A_d \) Diffuser Area

\( A_t \) Turbine Area

\( B \) Blade Number

\( C_P \) Power Coefficient

\( C_T \) Thrust Coefficient

\( D \) Fin Drag

\( F \) Prandtl Tip Loss Factor

\( F_x \) Force Obtained From the Load Cell in the x-direction

\( F_y \) Force Obtained From the Load Cell in the y-direction

\( L \) Fin Lift

\( M \) Quarter Chord Moment

\( M_{\text{Drag}} \) Quarter Chord Moment Caused by the Drag Offset

\( M_{\text{Lift}} \) Quarter Chord Moment Caused by the Lift Offset

\( M_z \) Moment Read by Load Cell

\( P_o \) Total Pressure

\( P_a \) Ambient Pressure

\( P_{\text{atm}} \) Atmospheric Pressure

\( P_D \) Pressure at Diffuser Exit

\( P_{\text{stat}} \) Static Pressure

\( P_{\text{tunnel}} \) Tunnel Pressure
$P_v$  Vapor Pressure
$R$  Rotor Raidus
$Re$  Corrected Reynolds Number
$Re_k$  Roughness Reynolds Number
$Re_s$  Reynolds number based on $s$
$Re_t$  Transitional Reynolds Number
$Re_u$  Uncorrected Reynolds Number
$S$  Fin Wetted Area
$St$  Strouhal Number
$V$  Volume
$a$  Axial Induction Factor
$a'$  Tangential Induction Factor
$c$  Fin Chord
$c_{pD}$  Back Pressure Coefficient
$c_d$  Drag Coefficient
$c_{dcav}$  Drag Coefficient in a Cavitating Condition
$c_{do}$  Corrected Profile Drag Coefficient
$c_{dou}$  Uncorrected Profile Drag Coefficient
$c_{dnoncav}$  Drag Coefficient in a Cavitation Free Condition
$c_l$  Corrected Lift Coefficient
$c_{l_{cav}}$  Lift Coefficient in a Cavitating Condition
$c_{l_{max}}$  Maximum Lift Coefficient
$c_{l_{noncav}}$  Lift in a Cavitation Free State
$c_{lu}$  Uncorrected Lift Coefficient
$c_{m1/4}$  Corrected Quarter Chord Moment Coefficient
$c_{m1/4u}$  Uncorrected Quarter Chord Moment Coefficient
\( c_p \) Pressure Coefficient
\( c_{pmin} \) Minimum Pressure Coefficient
\( d \) Length Scale
\( f \) Frequency
\( g \) Acceleration due to Gravity
\( h \) Height of Water Column
\( h_t \) Test Section Height
\( k \) Roughness Height
\( l \) Load Cell Center Offset from the Chord Line
\( q \) Dynamic Pressure
\( q_u \) Uncorrected Dynamic Pressure
\( r \) Augmentation Ratio
\( s \) Distance From Leading Edge to Roughness
\( t \) Fin Thickness
\( u_\# \) Velocity through any surface
\( u_r \) Shear Velocity
\( v_\infty \) Free Stream Velocity
\( v_a \) Velocity at station \( a \) (Free Stream)
\( v_D \) Velocity at Diffuser Exit
\( v_k \) Velocity at Roughness Height
\( v_o \) Velocity Upstream of Turbine
\( v_{RMS} \) RMS velocity
\( v_{wake} \) Velocity in Wake of Turbine
\( x \) Load Cell Moment Center \( x \)-direction Offset from Quarter Chord Point
\( x_{TE} \) Distance Downstream from the Trailing Edge
\( x_{tr} \) Transition Point
\( y \) Load Cell Moment Center \( y \)-direction Offset from Quarter Chord Point
\( z \) LDV probe position.
Acknowledgments

A special thanks goes out to Arnold Fontaine, William Straka and Dennis McLaughlin for their continued guidance and support as well as all supporting faculty at the Garfield Thomas Water Tunnel for keeping everything on track and running. I’d like to also thank all members of the team from other institutions who include Matthew Barone and Erick Johnson from Sandia National Laboratories, Henry Shiu and Case vanDam from University of California-Davis and anyone else from Penn State who helped me at some point during my education here.

Family and friends also played an instrumental role in my time at Penn State and I would like to extend thanks to them. My mother, father and brother helped both financially and mentally during my education and I would like to especially thank them. The experience would not have been as fun or meaningful if it wasn’t for my excellent base of friends and I thank them as well.

I’d like to also thank the sponsor of this project which is the Department of Energy and Sandia National Laboratories through contract # 775396. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-AL85000. The project and technical managers are Matthew Barone and Erick Johnson.
Chapter 1

Introduction

1.1 Objectives

The purpose of this study is to quantify the hydrodynamic and cavitation performance of a hydrofoil designed specifically for marine hydrokinetic turbine applications. This project is sponsored by the DOE through Sandia National Laboratories. Sandia National Laboratories is designing a three bladed hydrokinetic turbine for use in tidal or river applications which will feature hydrofoils specifically designed for this application. The foil presented in this work is named the MHKF1-180s and the foil cross section can be seen in Figure 1.1.

Figure 1.1: Cross section of foil designed tested in this thesis for the Sandia marine hydrokinetic turbine.

Foils performance can be quantified both experimentally and computationally. This
work presents the results from experimental water tunnel tests conducted in the Pennsylvania State University Applied Research Lab 12 Inch Water Tunnel. In order to verify testing procedures, identical tests were run on the NACA 4412 foil due to the large amount of historical data to compare to. The University of California - Davis designed the foil and performed the computational simulations on this foil. The experimental results are compared to these computational simulations herein.

1.2 Motivation

Increasing pressure towards clean, carbon free energy and reduction of fossil fuel dependency has pushed the growth of the renewable energy sector in the past years. For example Jose Zayas [1] and a report produced by the U.S. Energy Information Administration [2] presented an overview of the Department of Energy’s Wind and Water Power Program Overview which stated that the nation’s goal was that 80 percent of the nation’s electricity would come from clean energy sources by the year 2035. It was also stated in this overview that of that 80 percent, 15 percent would come from water power. This push towards "green" energy has promoted the growth of many industries such as wind, solar, hydroelectric, geothermal, marine and bio mass just to name a few of the biggest and fastest growing technologies.

Marine energy encompasses energy produced by tides, ocean waves, river and ocean currents and is named marine hydrokinetic energy because it harvests the kinetic energy of the moving water and converts it to electricity. This technology is a young technology, and in the case of hydrokinetic turbine technology, got on its feet by deriving most of its knowledge base from research related to wind turbines, conventional hydropower and the offshore marine industry.

There are uses for MHK turbines other than producing power directly for the grid. Turbines placed in rivers near small villages in remote locations can be more economical and environmentally friendly when compared to the diesel generators currently used to
generate power. One such example of a non grid connected remote location that utilized MHK technology is the Race Rocks Ecological Reserve which is powered by a Clean Current tidal turbine [3]. Other uses may include creating hydrogen from the electricity via electrolysis or desalinating sea water to create fresh drinkable water for areas where potable water may be scarce.

Harvesting the power of tides and currents has several advantages when compared to wind. First and foremost, the density of water is approximately 829 times the density of air allowing for a much higher energy density compared to air for a given flow speed. Power density is defined by equation 1.1 and is dependent on the density of the fluid ($\rho$) and the velocity in which it flows ($v_\infty$).

$$PowerDensity = \frac{1}{2} \rho v_\infty^3$$ (1.1)

Another major motivator is the predictability of the flow in which the device operates. Whether operating in a tidal straight, river, or ocean current the flow will be relatively predictable under normal weather conditions. For example, tidal energy has been predicted accurately years in advance due to the knowledge of the positions of the Moon and the Sun[4]. Wind or solar energy in comparison can only be accurately predicted days in advance[5, pgs. 11-22]. This predictability is beneficial because the time and amount of electricity delivered to the grid are well known quantities which can help supplement the grid during peak hours.

Unlike Conventional hydropower such as dams and tidal barrages, Marine hydrokinetic energy does not flood the surrounding area and can be designed to minimize blockage in the waterway to navigation and wildlife passage. Also MHK devices require much less civil engineering and building material [6]. dams and tidal barrages work on the principle of a pressure head, meaning that they operate by inducing a flow from a pressure head created by a height difference. A MHK device harvests energy from the motion of the fluid much like a wind turbine. This has minimal effect on the surrounding
ecosystem, when compared to conventional hydro, and in most cases will not impinge on wildlife migration or ship navigation.

While operation in water does have advantages it also has significant disadvantages. As mentioned above, water is approximately 829 times more dense than air. This results in a greater power density, but also results in higher blade loading and much larger stress on the blades requiring more material to maintain structural integrity. This increase in structure results in an increase in weight as well as an increase in cost.

Another drawback of MHK turbines is installing and long term maintaining of a piece of equipment underwater is difficult and costly. In order to successfully install a MHK turbine, divers, cranes and large barges are necessary which drives up installation cost especially if the turbine is installed in a remote location. Additionally more complex seals, bearings and anti-corrosion paints/materials are necessary due to the underwater environment. Additional monitoring may be necessary as well to monitor the health of the machine but also to monitor the effect the machine has on the surrounding environment. Progress in this area relies heavily on technologies developed for the existing offshore industry.

Erosion and biofouling are also problems associated with the underwater environment. Biofouling occurs when organisms such as algae or barnacles grow on the surface of the blades. The mechanisms in which biofouling generate is well addressed in a work from Melo and Bott[7]. This deposition of material causes performance losses on the blade and can effect the cavitation performance of the turbine, both of which will be addressed later in this thesis.

1.3 Siting Options for MHK Technologies

Marine hydrokinetic turbines can be installed anywhere where the kinetic energy of a moving fluid can be converted into electrical energy through mechanical means. This includes tidal straights, rivers, ocean currents and anywhere with waves. This work
focuses on in-stream tidal energy therefore it does not include wave energy.

**Tidal**

The tides are caused by the gravitational pull of the Sun and the Moon on the Earth. The Sun has about half the gravitational effect of the Moon despite its much larger mass due to the large distance separating the Earth and Sun [4]. The Sun’s gravitational pull is constant as compared to the moon’s gravitational pull which varies monthly because the Moon revolves around the Earth. The Earth also rotates underneath the Moon resulting in hourly changes in the gravitational effect.

When the Moon and the Sun are aligned on either side of the Earth as seen in Figure 1.2a their gravitational effect is additive. This period of very high and very low tides is referred to as spring tide. When the Moon and Sun are at a right angle as seen in Figure 1.2b the gravitational effects interfere destructively causing periods of less drastic tidal changes called neap tide. There are two spring tides and two neap tides a month as the moon revolves around the earth and the tides change from low to high tide every 6 hours due to the rotation of the Earth with relation to the moon.

![Diagram of spring and neap tides](image)

(a) Spring Tides  
(b) Neap Tides

Figure 1.2: Diagram of spring and neap tides. [4].

When siting a tidal turbine, certain locations are more suitable than others due to
greater flow speeds. Locations where the land narrows to form a nozzle shape increases the speed of the tidal current and therefore increases the amount of possible energy to be harvested. Figure 1.3 shows a contour plot of the global tidal amplitude [8] in which the contours are in 10 cm increments. Some major tidal sites that are being studied and implemented for MHK turbines now include Western Europe, Eastern Canada, The Puget Sound and Eastern Australia.

![Tidal Map](image)

**Figure 1.3:** Tidal map showing the amplitude of the tide. The contours are divided into 10 cm segments. [8]

Black & Veatch compiled a resource assessment for Western Europe in which the top 10 sites that generated 80 percent of the possible total tidal resource were studied [9]. The Black & Veatch top 10 tidal resources in the UK can be seen in Table 1.1 along with the percentage of the total possible resource they account for.

The Pentland Firth area holds a large majority of the tidal potential as seen from the above table. A map of the area is presented in Figure 1.4.

In Canada, areas such as the Bay of Fundy are very well suited for tidal turbines because the Bay of Fundy has the largest tidal variations on Earth. The amount of water that flows through the bay of Fundy is equivalent to the sum of all the rivers in the world [11]. A map of the Bay of Fundy can be seen in Figure 1.5.
Table 1.1: The top 80 percent of all the tidal resources in the UK [9]

<table>
<thead>
<tr>
<th>Rank</th>
<th>Site</th>
<th>Individual (%)</th>
<th>Cumulative (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Pentland Skerries</td>
<td>17.9</td>
<td>17.9</td>
</tr>
<tr>
<td>2</td>
<td>Stroma, P. Firth</td>
<td>12.7</td>
<td>30.6</td>
</tr>
<tr>
<td>3</td>
<td>Duncansby Head, P. Firth</td>
<td>9.3</td>
<td>39.9</td>
</tr>
<tr>
<td>4</td>
<td>Casquets, Channel Island</td>
<td>7.6</td>
<td>47.5</td>
</tr>
<tr>
<td>5</td>
<td>S. Ronaldsay, P. Firth</td>
<td>7.0</td>
<td>54.4</td>
</tr>
<tr>
<td>6</td>
<td>Hoy, P. Firth</td>
<td>6.3</td>
<td>60.8</td>
</tr>
<tr>
<td>7</td>
<td>Race of Alderney, Ch. Is.</td>
<td>6.3</td>
<td>67.0</td>
</tr>
<tr>
<td>8</td>
<td>S. Ronaldsay, P. Skerries</td>
<td>5.3</td>
<td>72.3</td>
</tr>
<tr>
<td>9</td>
<td>Rathlin Island</td>
<td>4.0</td>
<td>76.2</td>
</tr>
<tr>
<td>10</td>
<td>Mull of Galloway</td>
<td>3.7</td>
<td>79.9</td>
</tr>
</tbody>
</table>

Figure 1.4: Pentland Firth area map. [10]

A drawback of siting a turbine in a tidal straight is that, although the tide predictably changes every 6 hours, there are periods of time where the velocity is below the cut in velocity of the turbine and therefore no power is being produced. This period of low flow speed is referred to as slack water.

The Pacific Energy Ventures, LCC put together a siting methodologies handbook for the U.S. Department of Energy [13]. This is a very comprehensive work that focuses heavily on the regulatory procedures for siting marine hydrokinetic devices. It covers the regulatory framework of Alaska, Washington, Oregon, California, Hawaii, Maine, Massachusetts, Rhode Island and Florida.
River

Siting a turbine in a river is different from a tidal application due to the unidirectionality of the flow. This eliminates the need for a mechanism to capture both directions of a tide. The flow velocity of rivers is generally more consistent than tidal straights because there is no periods of slack water. Kahn published a paper that summarizes the technology of River Current Turbines and gives insights on the pros and cons of river current turbines [14]. Sornes also summarizes all small scale river technology in the 0.5 to 5 kW power range [15]. The paper touches on augmentation and summarizes current companies and their unique technologies. Verdant power also produced a paper focusing on uni-directional river current turbines [16]. The focus of this work is to present the status of technology development in Canada, the current technology and technical parameters of devices, energy production estimates, cost of energy production and analysis of Canadian technologies with respect to international development.

Rivers in the United States that are utilizing hydrokinetic energy include the Yukon River and the Mississippi River. Specifically Hydro Green Energy has installed a turbine in the tail race of the Army Corps of Engineers Lock and Dam No. 2 in Hastings, MN [3]. The focus of this study was to quantify fish passage and mortalities through the turbine. Underwater Electric Kite also operated a turbine in the Yukon River under the Alaska
Power and Telephone Company [17] in 2005. This project is focused on delivering power to a remote area where power is scarce. This project is the only project currently active in the Yukon, but there are many active projects in the Mississippi River. To view a list of these current projects see Reference [18] as well as installed a turbine at the Chitokoloki Mission in Zambia Africa in 2007. Figure 1.6 is a map showing the project locations on the Mississippi River.

![Mississippi River Hydrokinetic project map.](image)

**Figure 1.6: Mississippi River Hydrokinetic project map.** [18]

**Ocean Current**

Ocean currents, like rivers, are unidirectional in nature. Specifically, the Gulf Stream off the coast of Florida has been studied due to the large amount of energy potential and proximity to the population of Miami [19]. The Gulf Stream off the coast of Florida can reach velocities of 2 m/s as illustrated in Figure 1.7 by the black dots.
Figure 1.7: Image of the Gulf Stream including the Florida Current. Black dots refer to velocities greater than 1 m/s. [20]

The reference by the Pacific Energy Ventures, LCC [13] also covers siting and regulatory information regarding turbines installed in ocean currents such as the Gulf Stream.
1.4 Environmental Impacts

Several environmental studies have been conducted investigating the effects of installing, operating and maintaining a MHK turbine. Devine Tarbell & Associates [21] prepared a report for the Electric Power Research Institute (ERPI) on the environmental and permitting issues for tidal power in North America. This document is organized by the effects on the environment from installation, operation & maintenance and decommissioning.

The University of Washington held a workshop in 2010 titled Environmental Effects of Tidal Energy Development [22]. This workshop addressed the environmental effects as two groups, stressors and receptors. Stressors were factors that occur as MHK systems are installed, operated and decommissioned. Receptors were the elements of the ecosystem affected by the stressors. The stressors and receptors are outlined in Table 1.2

Table 1.2: Environmental stressors and receptors according to [22]

<table>
<thead>
<tr>
<th>Stressor</th>
<th>Receptors</th>
</tr>
</thead>
<tbody>
<tr>
<td>Presence of devices: static effects</td>
<td>Physical environment: near-field</td>
</tr>
<tr>
<td>Presence of devices: dynamic effects</td>
<td>Physical environment: far-field</td>
</tr>
<tr>
<td>Chemical effects</td>
<td>Habitat</td>
</tr>
<tr>
<td>Acoustic effects</td>
<td>Fish: migratory</td>
</tr>
<tr>
<td>Electromagnetic effects</td>
<td>Fish: resident</td>
</tr>
<tr>
<td>Energy Removal</td>
<td>Marine mammals and seabirds</td>
</tr>
<tr>
<td>Cumulative effects</td>
<td>Ecosystem interactions</td>
</tr>
</tbody>
</table>

Verdant Power, who installed six turbines in the East River, released an environmental assessment in 2011 which gave a detailed explanation of the environmental impacts in the East River [23]

A workshop was also held by the U.S. Department of Energy called Hydrokinetic and Wave Energy Technologies Technical and Environmental Issues [24]. This workshop broke down the environmental effects based on the resource type such as tidal estuaries, near shore environments, offshore environments and canals or other engineered waterways.
The U.S. Department of Energy also provided a report to Congress about the potential environmental effects of MHK turbines [25]. This lengthy report goes into significant detail on the types of environmental effects caused by MHK turbines.

### 1.4.1 Impact on Aquatic Life and Local Habitat

Installing the turbine will likely have the largest effect on aquatic wildlife in the area of the turbine. Drilling the seabed for the support structure and laying power transmission lines will disturb the local ecosystem and may even kill species that are not very mobile [21]. Installing these structures into a tidal stream can also affect the water quality around the device by introducing more turbulence into the local tidal stream, increasing the amount of sediment suspended in the water [21, 22].

The city of Hastings Minnesota performed a study on fish strike through a turbine designed by Hydro Green Energy [26]. The turbine was installed in the tailrace of the Army Corps of Engineers Lock and Dam No. 2 hydroelectric power plant located on the Mississippi river. The study was conducted by tagging 502 different sized fish with balloons and radio tags in which 402 fish were released through the turbine and 100 were released around the turbine. Once sufficiently downstream the balloons were inflated and the fish were collected for data processing. Out of the 402 fish released through the turbine, only one showed evidence of physical harm caused by the turbine. After closer examination it was concluded that the damage occurred because the balloon inflated prematurely causing the fish to pass through the turbine in a way that would never occur naturally. The conclusion of this study was that the single MHK turbine was very fish friendly and did not present a significant fish danger [27].

Increased or decreased predation is also a concern when installing a MHK device. The increased sonic level of the surrounding area of a turbine being installed or in operation can cause species of migratory aquatic life to avoid the area [21]. The increased noise, vibrations and structures can also attract certain species of animals to the area and increase predation [22]. Electromagnetic radiation from the generator and transmission
lines can also cause changes in behavior, especially in cartilaginous fishes [21].

The purpose of the turbines is to remove energy from the flow. This in turn slows the speed of the current, effects the turbulence level of the flow and, if surface piercing, changes the reflection and diffraction of waves [21]. This blockage however is much less severe than the blockage associated with a tidal barrage such as the barrage on the Rance River in France. A tidal barrage spans the entire width of an estuary capturing the high tide and then releasing it during low tide to create a low head tidal turbine. This creates obvious navigation and fish migration concerns as well as a large amount of civil engineering needed to create these barrages [6].

Navigation concerns arise when the turbine pierces the surface or the keel line of the largest ship is close to the height of the largest turbine. A work prepared by Pacific Energy Ventures LCC [13] for the U.S. Department of Energy Outlines siting methodologies for hydrokinetics. The document specifically goes over federal regulatory frameworks and the authorization process. Attractive tidal locations within the United States are also examined in detail such as Alaska, Washington, Oregon, California, Hawaii, Maine, Massachusetts, Rhode Island and Florida [13].

1.5 Existing Marine Hydrokinetic Devices and Designs

This section will be an overview of the existing MHK devices being designed, tested, or installed in the field. Several people have put together similar summaries. This thesis will focus on horizontal axis and will be broken down into open rotor and ducted devices. While other forms of turbines exist such as vertical axis turbines and oscillating hydrofoils they are beyond the scope of this work. The support methods used will also be outlined in this work.
1.5.1 Support Methods

The method of installing the turbine unit in the flow depends largely on the conditions and the design of the turbine. Unlike most modern day commercial wind turbines which use a monopile design, there are many different methods of installing a MHK turbine in a flow. Orme and Masters[28] analyzed several different mooring strategies and provided several pros, cons and general conclusions which are seen in the following sections.

Pile Mounted

Like mentioned above, the modern wind turbine utilizes this method of support. A pile would be first driven into the seabed at a predetermined depth and then the turbine would be installed on top. From Orme and Masters[28] 1.8 shows an example of a monopile supported MHK turbine.

Figure 1.8: Example of a monopile mounted MHK turbine[28]

The stiff structure of the monopile is very robust and can withstand the high loads associated with large utility scale devices in a marine flow environment. Another advantage is it is installed on the sea floor so it is of no concern to navigation, has no visual impact and, if deep enough is not subject to wave interaction. This method of support can also be installed in deep environments and the length of the pole can be made so the turbine is sufficiently above the seafloor to minimize boundary layer effects.

The fact that the monopile support method is completely submerged also introduces
difficulty performing minor maintenance on the device because the entire nacelle would have to be detached and brought to the surface which requires precise positioning of the support ship or the use of divers to do the maintenance underwater all resulting in increased maintenance costs. Depending on the location of the turbine, the maintenance ship could be costly. Also surface markers would need to be present since there are no other surface components and all the electronics are submerged requiring quality seals and bearings. This type of support also requires an active yawing system to align the turbine with the incoming flow. This active yaw system would likely yaw the entire nacelle into the flow, pitch the blades 180 degrees or some combination of both. This yaw system adds complexity, weight and cost to the entire MHK system.

A variation of the monopile support method is the telescopic monopile which is the same as a standard monopile but the nacelle can be brought to the surface using buoyancy and a telescoping pole. This adds the advantage of being able to maintain the turbine at the surface and adds the disadvantage of more complex design and higher loads when in the telescoped position. This method is also limited to depth.

Anchored and Gravity Bases

Anchoring a structure to the seafloor and using a gravity base have the same benefits and disadvantages with the exception that a gravity base does not drill into the seabed eliminating the environmental concern. Not drilling requires the base to be heavy enough to not move under extreme conditions based on the drag created by the unit. This may result in a very heavy structure and require a lot of material. In both cases there is a base structure, like a tripod, that has the nacelle and turbine attached to it. An example of this is the swan turbine\[29\] which can be seen in Figure 1.9

Another more unique method of anchoring a MHK device involves an anchored base and a tether, such as a chain, to support a buoyant nacelle and turbine. This is presented by Orme and Masters\[28\] and is summarized as follows and can be seen portrayed in Figure 1.10\[28\]. This method of support requires less structure and is mainly dependent
on a chain which reduces the overall cost. This chain can be varied in length to accommodate different depths. This support method has the added advantage of not requiring an active yaw system because the tether allows the turbine to align itself with the flow. In this method the unit is still submerged so it has all the benefits of being below the surface such as no visual impact, low navigation impact and no wave interaction.

The buoyant tether support system is again completely submerged so it has all the disadvantages associated with pile mounted devices such as additional surface markers need to be present, minor maintenance becomes difficult and all the components are submerged underwater requiring complex seals and bearings. In addition to this, the self aligning feature of this support method also presents a disadvantage. In order to
self align, the turbine will always be in the downstream position, meaning that the flow will always encounter the nacelle before going through the turbine. This causes non-uniformities in the flow which can cause vibrations and performance decreases. Also the area in which the turbine sweeps during its aligning with the flow can be quite large and can lead to a decreased number of turbines in a given area when compared to other support methods. Since the unit is allowed to drift back and fourth as the flow direction changes there is also a risk of the tether getting tangled.

**Floating Moored**

A floating moored support structure supports the turbine underneath a floating device that is tethered to the sea floor or an existing structure like a bridge abutment. This method of support is explained in the 2009 proceedings of the International Ship and Offshore Structures Congress[30] and is summarized as follows. This method of support benefits from the simplicity and reduced installation and maintenance cost of not being completely submerged in water. There is no need for additional navigation markers and it can be used in waters of any depth such as shallow waters closer to the coast. Also the generator and electric equipment can be housed above water reducing the complexity of the waterproofing necessary to keep the system dry.

![Figure 1.11: Example of a floating moored MHK turbine][31]

Since this support structure is surface piercing it does become a navigation hazard and may be visually unappealing to some. Floating also makes the structure susceptible
to waves and severe weather which can damage the structure and turbine. Also since
the unit is allowed to drift back and fourth as the flow direction changes there is a risk
of the tether getting tangled.

**Sheath System**

This mooring method addressed by Orme and Masters\cite{28} and summarized here utilizes
a mechanical system that raises and lowers the nacelle and turbine along one or more
piles driven into the seabed. This is different from the monopile design because there is
always structure piercing the surface and there may be more than one pile per turbine.
This method can be seen in Figure 1.12\cite{28}.

![Figure 1.12: Example of a sheath system MHK turbine\cite{28}](image)

This system allows the turbine to be raised to the surface for maintenance purposes.
Also since there is one or more monopiles the structure is robust and can withstand large
loadings. Since there is structure that is surface piercing all the electronic systems can
be stored above water and there is no additional surface marking needed.

Marine navigation, visual obstruction and subject to waves and weather are all dis-
advantages of having a surface piercing system. Along with this the installation cost
will be increased due to the complexity of the mechanical hoist system necessary to raise
and lower the turbine into position. Proper alignment of this unit with the average flow
direction is paramount for this method because the nacelle cannot be yawed making it
a better option for river or current applications. This then requires the unit to either
have symmetric or variable pitch blades if installed in a tidal straight to capture both the ebb and flood of the tide.

### 1.5.2 Horizontal Axis Devices

Horizontal axis turbines are turbines which the axis of rotation is parallel to the direction of incoming flow. Most of the MHK horizontal axis designs are strongly based on wind turbine technology because it is a highly refined and optimized design. However, you cannot simply scale down a wind turbine and install it in a tidal stream. The first reason is the loads are much greater on the blades due to the increased density of water. Also, wind turbine airfoils are designed to have a leading edge low pressure peak in order to reduce the sensitivity to surface roughness. This pressure peak is a poor design choice for MHK turbine blades due to the concern of cavitation. Another issue is that water (especially salt water) is very corrosive and damaging to mechanical parts present in a turbine and generator. Designers pulled ideas from the marine industry using high quality seals and bearings already present in existing marine technologies to help prevent such corrosion.

A single rotor can be installed per unit or several rotors can be fixed to the same support structure which adds additional benefits. The use of a duct, diffuser or both can also increase the overall efficiency of the device and is addressed in a later section.

### Open Rotor Designs

Wind turbines and open rotor horizontal devices share a lot of similarities. They maintain the same general blade shape of chord decreasing from root to tip as well as being highly twisted at the root and washing out at the tip. Though there are many similarities between open horizontal MHK turbines and wind turbines, there are several choices a designer is faced with when designing a marine hydrokinetic device. The most critical choices are fixed or variable pitch blades, gearbox or direct drive generator, and the method used to align the turbine with the oncoming flow. As comparison, most
commercial wind turbines today are variable pitch, mechanical nacelle yawing turbines. Commercial wind turbines generally use a speed increasing gearbox to run the generator but there is a growing trend in using direct drive generators[32].

Fixed pitch blades benefit from being mechanically simpler due to the lack of a motor, complex seals and bearings used to pitch the blades. Fixed pitch blades are also lighter, easier to maintain, and cheaper than the variable pitch counterpart. Without seals and bearings there is less chance for something to become damaged due to leaking seawater. Over speed protection in fixed blade rotors is done by stall regulation. The blades are designed to stall at a certain flow speed around the maximum expected current speeds under normal operating conditions[33]. Stall regulated blades do need additional over speed protection in the form of a powerful mechanical break. This mechanical break is needed in extreme conditions, such as bad weather where the stall regulated blades are not enough. There are situations where a break may not be needed however. If the turbine is mounted deep enough to avoid the effects of weather then a break may not be necessary. Also the tides are much less variable than the winds, so extreme situations are rather unlikely depending on the depth of the installed unit. If the unit is installed deep enough it is not subject to surface weather, but if the unit is floating or surface piercing it will be subject to extreme cases caused by weather.

Variable pitch blades have more mechanical complexity due to seals and bearings, but do have several benefits. Variable pitch blades are able to maintain the optimum angle of attack with varying flow speeds by pitching the blades increasing the efficiency compared to fixed pitch blades. This however requires a control system which adds additional complexity, cost and weight. Over speed protection is done by feathering the blades into the flow. When feathered the blades no longer produce lift or torque and therefore the turbine does not rotate. The ability to pitch the blades 180 degrees has an added benefit in that it allows both directions of the tide to be captured and gives the option of not having to yaw the nacelle in tidal applications. This reduces the complexity of the overall system so even though using variable pitch blades adds complexity and
cost, the ability to not have to yaw may make up for this in the end.

Like wind turbines, some hydrokinetic turbines mechanically yaw into the incoming flow and have variable pitch blades to maintain optimal loading at different flow speeds. These two features are well suited for wind turbines, but may introduce an added complexity and maintenance issue when applied in an aquatic environment. When installed in a tidal stream, it is necessary to be able to capture both the ebb and flow of the tide. Mechanical and passive yawing of the nacelle, variable pitch blades, symmetric blades, opposite facing rotors, and tethered mooring system all allow the unit to be aligned with the incoming flow. Multiple rotors on a single unit present some installation benefits as well. For example if the unit is pile moored, then only one pile has to be driven into the seabed for multiple turbines on that pile.

Single open rotor devices are the simplest mechanically when compared to the other types of turbines. They do not have a duct or multiple turbines on a unit which complicates and adds structure. They are the closest resemblance of wind turbines, and in some cases are simply scaled down wind turbines. Some examples of single open rotor designs can be seen in Table 1.3.

<table>
<thead>
<tr>
<th>Technology</th>
<th>Diameter</th>
<th>Rated Power</th>
<th>Support Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Atlantis - AR1000</td>
<td>18 m</td>
<td>1 MW</td>
<td>gravity</td>
</tr>
<tr>
<td>Bourne Energy - River Star</td>
<td>6.1 m</td>
<td>.05 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Hammerfest Strom - hs1000</td>
<td>10 m</td>
<td>1 MW</td>
<td>gravity</td>
</tr>
<tr>
<td>Oceanflow 2400 - Evopod</td>
<td>16 m</td>
<td>1.5 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Swan Turbine LTD. - Swan Turbine</td>
<td>-</td>
<td>1-3 MW</td>
<td>anchored</td>
</tr>
<tr>
<td>Tidal Generation - Deep-Gen</td>
<td>18 m</td>
<td>1 MW</td>
<td>anchored</td>
</tr>
<tr>
<td>Tocardo Tidal Energy - T500</td>
<td>10 m</td>
<td>0.5 MW</td>
<td>anchored</td>
</tr>
<tr>
<td>Verdant Power - FFS</td>
<td>5 m</td>
<td>.035 MW</td>
<td>monopile</td>
</tr>
<tr>
<td>Voith Hydro</td>
<td>16 m</td>
<td>1 MW</td>
<td>gravity</td>
</tr>
</tbody>
</table>

Three examples of horizontal open rotor turbines can be seen in Figure 1.13. The major difference between these turbines is the way they orient themselves to capture both the ebb and flow of the tide. The Evopod and verdant turbine use a passive yawing
method in which the flow pushes the turbine into the correct position. The swan turbine uses a mechanical nacelle yawing mechanism to align itself with the flow.

(a) Evopod Turbine  
(b) Swan Turbine  
(c) Verdant Turbine  

Figure 1.13: Horizontal open rotor hydrokinetic devices [34, 29, 35]

Of the turbines introduced in Table 1.3, the Evopod[34] and the Riverstar[36] are housed underneath a floating platform. These designs come with all the advantages and disadvantages of a floating moored support system as described previously.

The Evopod is kept afloat with buoyant hydrofoil shaped members that also aide in aligning the turbine with the flow. The Evopod is a 16 meter, 3 bladed, fixed pitch turbine rated at 1.5 MW. A 1/40th model of the turbine was tested in a flume at Newcastle University. The company website boasts a high survivability in extreme weather[34].

The Riverstar has no yawing capability because it is designed for unidirectional applications. It features a 6.1 meter diameter rotor that generates 50 kilowatts of power at a designed rotor speed of 12 RPM. The Riverstar is designed so that multiple units can be tethered together and the tops of the units that stick out of the water can be blended into the surroundings with grass, rocks or sand[36].

The Atlantis - AR1000[37], Hammerfest Strom - HS300[38], The Swan Turbine[29], Tidal Generation’s Deep-Gen [39] and Voith Hydro[40] all utilize an anchored or gravity base mooring method as outlined in Section 1.5.1.

The AR1000, Swan Turbine and Tidal Generation Deep Gen use a mechanical nacelle yawing system to align itself with the flow. The Swan Turbine can be seen in
Figure 1.13b. This type of yawing technology is taken directly from the wind energy industry. Most commercial sized wind turbines use a mechanical yaw system to align into the wind. These yawing mechanisms require some sort of control system to detect when the flow direction changes and rotate the turbine accordingly. The addition of a mechanical yawing mechanism assures that the turbine is aligned with the incoming flow at all times, but adds mechanical complexity due to the motors, bearings, seals, and structure needed to support the yawing system. Also lubrication for the bearings and seals can leak into the surrounding water which is an environmental hazard.

Rolls-Royce bought out Tidal Generation LTD[41]. This turbine has an 18 meter diameter rotor with 3 variable pitch blades. The turbine has variable pitch blades in order to increase efficiency and a mechanical yaw nacelle to align itself with the flow. Both of these mechanical systems add a lot of complexity and weight to the system. The full scale turbine is rated at 1 MW but a 500 kW version has been tested at the EMEC site in Orkney Scotland[39].

The Hammerfest Strom Tidal Stream Turbine [38] is equipped with variable pitch blades to capture both directions of the tidal current by pitching a full 180 degrees. This means that the nacelle can remain fixed and does not need to yaw into oncoming flow. The problems with this type of design is that the turbine must be very carefully placed so that it is normal to the flow. This turbine features a 10 meter diameter rotor rated at 300 kW. There was an environmental study, performance modeling and site survey performed on a 10 MW site. A utility scale 10 MW array is planned for the Sound of Islay, Scotland[38].

The last turbine to be mentioned in Table 1.3 is the Verdant Power turbine[35, 42, 16] pictured in Figure 1.13c. This turbine is a downstream turbine, meaning that the flow encounters the nacelle first then passes through the rotor. The fact that this turbine is a downstream turbine allows it to utilize a passive yaw system to align itself with the flow. As the flow changes direction it pushes the rotor in line with it much like a weather vane. This method of yawing is also relatively simple because no mechanical system
is needed. A bearing and seal is necessary for the union between the pile and nacelle however. Another drawback from a downstream turbine is that the flow that encounters the rotor is affected by the nacelle and tower. This causes non-uniformities in the flow and may effect the performance. Verdant power has installed 6 of these turbines into the East River in New York City. The project is called the Roosevelt Island Tidal Energy project or RITE project[43].

Mounting multiple turbines on the same support structure adds the benefit of only having to move the turbines at one point. This eliminates the need to drill multiple holes for piles or have several floating structures. This can lead to reduced area needed to operate the turbines and therefore more turbines can be installed at a given location. These structures that house the multiple turbines can be more complex than those that only support one turbine. Supporting multiple turbines also means the structure must be more robust. Table 1.4 shows existing devices that support more than one turbine per support structure.

Table 1.4: Multiple rotor horizontal axis devices

<table>
<thead>
<tr>
<th>Technology</th>
<th>Diameter</th>
<th>Rated Power</th>
<th>Support Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aquantis - C-Plane</td>
<td>40 m</td>
<td>2.6 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Atlantis - AK Series</td>
<td>18 m</td>
<td>1 MW</td>
<td>monopile</td>
</tr>
<tr>
<td>Bourne Energy - Tidal Star</td>
<td>6.1 m</td>
<td>0.05 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Hydra-Tidal - Morlid II</td>
<td>23 m</td>
<td>1.5 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Marine Current Turbines - SeaGen</td>
<td>20 m</td>
<td>1.8 MW</td>
<td>monopile</td>
</tr>
<tr>
<td>Scott Renewables - STT</td>
<td>12 m</td>
<td>2.4 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>SMD Hydrovision - TidEl</td>
<td>15 m</td>
<td>1 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>University of Strathclyde - CoRMaT</td>
<td>3 m</td>
<td>0.03 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Tidal Energy Ltd. - Delta Stream</td>
<td>15 m</td>
<td>1.2 MW</td>
<td>gravity</td>
</tr>
<tr>
<td>Statkraft - Tidevannskraft</td>
<td>-</td>
<td>1 MW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Tidal Stream Turbine</td>
<td>20 m</td>
<td>1 MW</td>
<td>anchored to seabed</td>
</tr>
</tbody>
</table>

The Atlantis AK Series [37] (Figure 1.14a), Bourne Tidal Star [36], Hydra-Tidal - MorlidII [44] and Statkraft Tidevannskraft [45] all use opposite facing rotors in their design. The Atlantis and Bourne, KESC turbines all operate with only one turbine spinning at a time. This allows the turbine to capture both the ebb and flow of the
tide without the complication of variable pitch blades or a nacelle yawing mechanism. This configuration can be moored by monopile like the Atlantis AK, or by a tethered floating platform like the Tidal Star. While one turbine is operating the other is in a brake state not spinning. The downfall to this is that the rotor that is not operating is causing enormous amounts of excess drag causing large stresses on the support member and causes added blockage effects further decreasing the efficiency of the turbine. The braked rotor also requires a powerful mechanical break to stop the rotor adding weight and complexity.

The fact that there are two rotors and only one is operating at a time may be an inefficient use of materials and resources. The Hydra-Tidal MorlidII and Statkraft Tidevannskraft uses two sets of counter rotating rotors for a total of 4 turbines per structure. Both of these devices are supported by floating structures that are tethered to the seabed or shore. The MorlidII can be seen in 1.14b. The difference between these devices and the others previously mentioned are that the blades can pitch so that all four rotors can be operating simultaneously. These rotors will still have an effect on each other, but they will still be generating power despite of this. The MorlidII turbine features four 23 meter rotors that produce a total of 1.5 MW of power. Each turbine has 2 blades made from laminated wood. This device has been tested full scale in August 2012 at Lofton Norway.

The CoRMat [46, 47] turbine developed by the University of Strathclyde [48] as seen in Figure 1.14c has two 2.5m diameter counter-rotating rotors that produce 42 kW of power. These counter-rotating rotors remove the swirl in flow generated by the rotation of the turbine blades and also eliminate the yaw reaction from rotation. This device has no gearbox and requires active hydraulics to match the torque of the two rotors. a 1/30th model has undergone testing in the University of Strathclyde's tow tank and a 1/10th model has undergone sea tests [46].

Tidal Stream’s Tidal Stream Turbine [49] can have up to six turbines mounted on a single fixture requiring only one drilling point in the seabed as opposed to six separate
drilling sites required for single devices. This turbine can be seen in Figure 1.15a. This unit has two buoyant spar buoys that support the arms that house the turbines and generators. This floating unit is attached to a rigid swing arm that is pinned to a base anchored to the seabed. This allows the unit to swing around with the changing of the tides. There would have to be some sort of reversal mechanism however to make sure the cables don’t get wound up. The floating spar buoys also make it easy to install. The unit is towed on sight behind a boat, anchored to the seabed, then the spar buoy ballasts are filled with water to submerge the unit to the desired depth. When maintenance is needed the unit can float to the surface and provide a stable working platform to perform the necessary and inevitable maintenance.

Scotrenewables STT turbine [50, 51] as seen in Figure 1.15b is housed underneath a floating metal tube that has two keels underneath it. These two keels fold out to support two 12 meter diameter rotors that deliver 1.2 MW of power. These keels can be folded up against the body of the tube in the event of bad weather or in order to tow in to location.

The SeaGen [52] as seen in Figure 1.16a is moored on a monopile. The monopile supports a cross arm which houses the turbines and generators. The turbines counter rotate in order to eliminate torque effects. An interesting feature of this design is that the cross arm is free to move up and down the monopile, making servicing the unit much
Figure 1.15: Examples of multiple turbines attacked to the same support structure [49, 50]

easier than the use of divers. The turbine captures the ebb and flow of the tide by the use of variable pitch blades. Marine Current Turbines also states that piles are the only practical choice for mooring of commercial sized hydrokinetic turbines due to the large thrust reaction, which is on the order of 100 tonnes per MW [52].

The Delta Stream turbine [53] pictured in Figure 1.16b is a modular base unit with three turbines attached to each point of the triangle. The base is hollow and light when it is on land, then fills with water to hold it to the seabed. The device features fixed blades and a mechanical yaw system to capture both directions of the tide.

Figure 1.16: Pile and gravity moored MHK turbines [52, 53]
The SMD Hydrovision Tidel turbine [54] and the Aquantis C-Plane [55] are buoyant units which are tethered to the seabed but do not puncture the surface of the water. These designs can be seen in Figure 1.17a, and Figure 1.17b respectively. The SMD Hydrovison turbine features two buoyant generators supported by a streamlined cross beam support. The unit has fixed pitch blades and changes direction by swinging around like most other floating tethered turbines. SMD Hydrovison states in their website [54] that the unit is designed to be installed at depths below 30 meters to eliminate the vertical velocity gradient and therefore operating under more uniform loads. Also at this depth biofouling and cavitation will be less of an issue due to the lack of light and high pressures respectively.

The C-Plane has two 40 meter counter-rotating turbines generating 2.3 MW of power mounted on a wing structure. Like the Cormat, the counter rotation negates any torque effects. this design features a flight control mechanism to change depths based on varying current and can also dive to depth where weather does not affect the flow. The design also does not require a direction change mechanism because it is designed to operate in the uni-directional flow of the Gulf Stream.

Figure 1.17: Positively buoyant MHK turbines tethered to the seafloor [54, 55]
Augmented Rotor Designs

Augmenting a rotor with a shroud or duct is a common design feature in Marine hydrokinetic devices. A duct or shroud is a nozzle, diffuser or a combination of the two. Since tidal straights are bi-directional, often a symmetric duct is used so the turbine does not have to realign itself with the oncoming flow. In this scenario symmetric blades must be used in order to capture both directions of the flow. There has been previous research performed on the use of ducts on wind turbines [56, 57, 58], however there is rarely ever a duct implemented on a utility scale wind turbine because wind turbines of this scale are on the order of 70 meters in diameter and sit 80 meters off the ground on a tower. The addition of an enormous duct would add a large amount of weight which would require a more robust tower and yawing mechanism. Also wind varies frequently in both speed and direction requiring the turbine to yaw into the flow frequently requiring more robust hardware to yaw the turbine. The addition of the duct would also add a large drag force putting more stress on the turbine as well as cause a blockage which would divert air around the turbine. Tidal turbines on the other hand do not share all of these disadvantages. Tides, like stated previously, are very predictable and change direction about every 6 hours requiring less yawing. Also the duct can be designed to be neutrally buoyant so the weight of the duct does not have as much of an effect on the tower structure, but the tower will still be affected by the large amount of drag introduced by the large surface area of the duct which will still cause blockage diverting fluid around the turbine.

In a paper by Kirke 2006[59] he states several advantages of a duct or diffuser. A diffuser protects the turbine from debris that can damage the blade tips and also acts as a safety mechanism for divers or fish so they are not struck by the rotating blades. The duct also shades the turbine blades from sunlight which inhibits the growth of biological life. Tip losses are reduced therefore increasing the efficiency. The addition of a duct also reduces sensitivity to flow directionality by straightening the flow as it enters the duct [60].
A nozzle works on the principle of conservation of mass, and essentially funnels water through the turbine at a greater velocity than the free stream velocity due to a reduction in area. Conservation of mass states that as area decreases velocity must increase to insure the same mass flow. Conservation of energy can be seen in the formula below:

\[ \iiint \frac{\partial \rho}{\partial t} dV + \iint \rho \mathbf{u} \cdot \mathbf{n} dS = 0 \quad (1.2) \]

If the flow is steady the left term is neglected, and in the simple case of a nozzle or diffuser there are only two surfaces with mass flux through them. The equation can then be simplified to the following:

\[ \int_{A_1} \rho_1 u_1 dA = \int_{A_2} \rho_2 u_2 dA \quad (1.3) \]

Since power scales with the velocity cubed, even a modest increase in velocity will cause a significant increase in power compared to an unducted turbine of the same diameter if the power is calculated using the turbine area as opposed to the diffuser exit area.

Since the duct is often symmetrical, the other side acts as a diffuser. The addition of a diffuser has been extensively studied and tested on both wind and water turbines. Igra [61, 62, 56] has several papers on experimental and theoretical diffuser augmented turbine work and claim that for the same diameter turbine under the same conditions, the power can be increased by a factor of 2 with the use of a duct. This result however normalizes the power with the turbine diameter and not the diffuser exit area and makes the case that with a duct, the same power can be produced using a smaller diameter. Kirke [59] summarizes the most recent developments in ducted turbine technology. Setoguchi [63] has a work in which a design is presented for a two way diffuser for fluid flow energy conversion systems. Gaden and Bibeu [64] performed a numerical study on the effects of a diffuser and came to the conclusion that 3.1 times the power can be generated by using a diffuser when compared to a conventional horizontal axis turbine with the
same turbine area. This claim however has never been proved in practice.

van Bussel [57] published a paper where several experiments involving ducted turbines were compared and in these experiments a large power increase was claimed when compared to unducted turbines. The conclusions drawn by this work were that the prior work was affected by large blockage effects and unrealistic increased performance benefits. Also when the results were recalculated by normalizing the power coefficient with the diffuser exit area instead of the turbine area it was found that no device exceeded the Betz limit. Also a conclusion was drawn that in practice the performance will be lower due to viscous effects and separation at the diffuser surface [57]. The main point to be taken away from this is that the performance number were inflated due to large blockage effects and non-dimensionalizing by the turbine area instead of the diffuser exit area.

The physics of a diffuser are based on momentum theory and are outlined in works by Foreman et. al. [60], van Bussel [57] and Werle et. al. [65]. A diagram from reference[65] shows an example diffuser and the nomenclature used as seen in Figure 1.18

A diffuser works by entraining more fluid through the turbine than normally would pass through if the diffuser were not present. According to van Bussel [57], the achievable
The power of a diffuser augmented turbine is comparable to a normal turbine with the same diameter as the diffuser exit, but larger performance gains result from a low back pressure at the diffuser exit. It has been determined by van Bussel however that these large back pressures that were seen experimentally were due to large blockage effects caused by the large duct in a wind tunnel [57]. This back pressure creates a larger pressure gradient through the turbine therefore augmenting the flow. This back pressure is caused by the flow being diverted in the radial direction due to the Kutta condition resulting in a difference in velocity and pressure at the exit of the diffuser when compared to the free stream. The velocity ratio $\gamma$ between station $a$ and $D$ of Figure 1.18 is expressed below:

$$\gamma = \frac{v_D}{v_a}$$  \hfill (1.4)

The back pressure coefficient at the end of the diffuser is defined as:

$$c_{pD} = \frac{P_D - P_a}{\frac{1}{2}\rho v_a^2}$$  \hfill (1.5)

which can be expressed in terms of $\gamma$ by applying Bernoulli’s principle as seen below:

$$c_{pD} = 1 - \left(\frac{v_D}{v_a}\right)^2 = 1 - \gamma^2$$  \hfill (1.6)

According to Werle et al. [65], the augmentation ratio (defined as $r$) is the ratio of the power generated with a diffuser over the power generated without a diffuser assuming the same turbine diameter and equal freestream velocity. This ratio is a function of the back pressure $c_{pD}$, turbine to diffuser area ratio $\frac{A_t}{A_D}$ and diffuser efficiency $\eta_d$. The expression for the augmentation ratio produced by a diffuser can be seen in the equation below:

$$r = \frac{1}{2} \sqrt{\frac{(1 - 9c_{pD})}{(1 - \eta_d + \eta_d \frac{A_t}{A_d})}}$$  \hfill (1.7)

In this equation $A_t$ is the turbine area and $A_d$ is the diffuser area. Equation 1.7 can
be seen plotted in Figure 1.19 over a range of back pressure coefficients and area ratios. In this plot it is assumed that the diffuser is 85% efficient.

![Figure 1.19: Power augmentation caused by a duct. The blue dotted line corresponds to the exit pressure coefficient of a horizontal axis turbine without a duct.](image)

From Equation 1.6 it can be seen that the more negative $c_{pD}$ (back pressure) the more augmented flow. This negative back pressure coefficient corresponds to an increased velocity through the turbine when compared to a normal unducted turbine with the same turbine area, resulting in an increase in RPM and reduction of the disc loading. The increase in RPM also benefits the drive train and allows a smaller gearbox to be used to transfer power and the decreased disk loading reducing the cost of the rotor [60].

In Figure 1.19 the augmentation ratio intersects the $\frac{A_D}{A_D} = 1$ line at a $c_{pD}$ value of -.325. This corresponds to a standard horizontal axis turbine with no duct. It can be seen for the same back pressure but increasing the duct area the power increases by 17 and 50 per cent for area ratios of 1.5 and 3 respectively.

In summation, there are theoretical benefits to implementing a duct on the turbine, but the performance benefits of such a design have not yet been proven in practice. The performance values are often larger than they should be due to non-dimensionalizing by the turbine area as opposed to the exit diffuser area as well as large blockages in tunnel
tests. Several developers have designed turbines that incorporate a duct. Some of these companies can be seen in the Table 1.5.

Table 1.5: Augmented horizontal axis devices.

<table>
<thead>
<tr>
<th>Name</th>
<th>Diameter</th>
<th>Rated Power</th>
<th>mooring method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alstom Hydro - Clean Current Turbine</td>
<td>17 m</td>
<td>2.2 MW</td>
<td>gravity/pile</td>
</tr>
<tr>
<td>Atlantis Resource Corp. AS Series</td>
<td>variable</td>
<td>.1, .5, 1 MW</td>
<td>pile, gravity, float</td>
</tr>
<tr>
<td>Free Flow Power - FFP Generator</td>
<td>2 m</td>
<td>10 kW</td>
<td>anchored</td>
</tr>
<tr>
<td>Hydro Green Energy - Kensington</td>
<td>3.66 m</td>
<td>250 kW</td>
<td>floating tethered</td>
</tr>
<tr>
<td>Lunar Energy - Rotech Tidal Turbine</td>
<td>11.5 m</td>
<td>1 MW</td>
<td>gravity</td>
</tr>
<tr>
<td>Open Hydro - Open Centre Turbine</td>
<td>10 m</td>
<td>1 MW</td>
<td>anchored</td>
</tr>
<tr>
<td>UEK Corp. - Underwater Electric Kite</td>
<td>2-5 m</td>
<td>1 MW</td>
<td>floating tethered</td>
</tr>
</tbody>
</table>

The Alston Hydro Clean Current Turbine [3] (Figure 1.20a), Atlantis AS Turbine [37] (Figure 1.20b), Lunar Energy Rotech Tidal Turbine [66] (Figure 1.20c) and Open Hydro Open Centre Turbine [67] (Figure 1.21) are all symmetrical ducted turbines with bi-directional blades. These turbines do not have the ability to yaw therefore positioning the devices normal to the flow is important. Of these four technologies, all but the Lunar Energy device utilize a permanent magnet generator which is the only moving part on the device. The Lunar Energy devices uses high pressure oil to drive a generator. The Lunar Energy turbine also incorporates a “cassette” feature in which the turbine can be slid out of the duct like a cassette for maintenance purposes. The Clean Current Turbine and Open Centre Turbine feature an open center section of the turbine to allow for fish passage. A 5 meter diameter 65 kW version of the Clean Current Turbine is currently installed at the Race Rocks Ecological Reserve in British Columbia, Canada. All of these devices are moored using a gravity base or monopile with the exception of the Lunar Energy turbine which is moored using exclusively a gravity base.

The Hydro Green Energy Kensington turbine [26] (Figure 1.22a), Under Water Electric Kite [17] (Figure 1.22b) and Free Flow Power FFP Generator [68] (Figure 1.22c) are all asymmetrically ducted devices that are designed for river, current or any other uni-directional flow application. The FFP generator has a permanent magnet generator
and only one moving part. It is designed to be anchored to existing structures such as bridge abutments. The Underwater Electric Kite features a side by side design with an augmentor ring which causes a low pressure region behind the turbine augmenting the flow through the device. The device is positively buoyant and is moored with a tether. The angle of attack of the device can be adjusted using ballast control. This device is currently being tested in the Yukon River in Alaska. The Kensington turbine is designed to be installed in clusters. It is suspended along the surface of the flow using tethers and anchors.


1.6 Blade and Foil Design

1.6.1 Blade Element Momentum Theory (BEMT) and the Betz Limit

Blade element momentum theory is the method that is used to design marine hydrokinetic turbines. This theory is also used in wind turbines, helicopter rotors and propellers. Blade element momentum theory is a combination of momentum/actuator disk theory and blade element theory. A very thorough derivation and explanation of momentum and blade element momentum theory is given by Manwell [69].

Actuator disk theory treats the rotor as an instantaneous pressure drop and does not account for the blade design. This pressure drop across the rotor causes some of the flow to be diverted around the turbine instead of through it. The larger the pressure drop, the more power is being harvested and the more flow is diverted around the turbine. This can be seen in Figure 1.23.

The pressure drop across the actuator disk is derived using Bernoulli’s equation before and after the rotor and is defined in equation 1.8. The velocities can be expressed as a function of the axial induction factor defined in equation 1.9. The thrust and the power coefficients can be expressed as a function of this axial induction factors and can be seen in equations 1.10 and 1.11 [69]. Taking the derivative of the power and setting that equal to zero gives the maximum value for $a$, which is 1/3. Plugging that back into equation 1.11 gives the maximum value for $C_p$ which is $\frac{16}{27}$, otherwise known as the Betz limit.
Figure 1.23: Diagram showing the stream tube pressure drop and velocities at several stations. [70]

\[ \Delta P = \frac{1}{2} \rho (v_o^2 - v_{wake}^2) \] (1.8)

\[ a = \frac{v_o - v_{wake}}{v_o} \] (1.9)

\[ C_t = 4a(1 - a) \] (1.10)

\[ C_p = 4a(1 - a)^2 \] (1.11)

Using momentum theory, the optimal chord and twist distributions can be found using equations 1.12 and 1.13 [69]. In this analysis twist is defines as the angle of the chord line with respect to the rotor plane.

\[ c = \frac{8\pi rsin\phi}{3Bc_t\lambda_r} \] (1.12)
\[
\theta = \tan^{-1}\left(\frac{2}{3\lambda_r}\right)
\]  

(1.13)

The assumptions of bladed element momentum theory are it is two-dimensional, steady, includes root and tip losses and does not account for radial flow along the span of the blade [69]. It treats the blade as many individual blade sections and is based off the actual blade geometry as opposed to actuator disk theory which ignores the blades. An example of a blade section can be seen in Figure 1.24. In this figure \(\alpha\), \(\theta\) and \(\phi\) refer to the angle of attack, twist angle and flow angle respectively. Also \(\omega\) is the rotor speed at a specific radial station and \(a'\) is the angular induction factor defined in equation 1.14. The addition of rotation and the actual blade geometry also introduces the important term referred to as the tip speed ratio, which is the ratio of the velocity of the blade tip to the free stream velocity as seen in equation 1.15. The power coefficient for BEMT can be seen in equation 1.16. In this equation \(\lambda_h\) is the tip speed ratio at the hub, \(\lambda_r\) is the tip speed at the local blade section, \(F\) is the Prandtl tip loss factor and \(\left(\frac{c_l}{c_d}\right)\) is the inverse lift to drag ratio.

Figure 1.24: Blade section showing the velocity triangle [71].

\[
a' = \frac{\omega}{2\Omega}
\]  

(1.14)

\[
\lambda = \frac{\omega R}{v_o}
\]  

(1.15)
\[ C_p = \frac{8}{\lambda^2} \int_{\lambda_h}^{\lambda} F \lambda^3 a'(1 - a') \left( 1 - \left( \frac{c_d}{c_l} \right) \cot \phi \right) d\lambda_r \]  

(1.16)

1.6.2 Design Criteria

When designing a turbine rotor, several things must be considered which include the number of blades, the twist and taper distributions, the foil shapes, and tip speed ratio. Design of a turbine blade is often broken up into three sections which are the root, mid-section and tip where each of these three sections has a different foil shape. Each section requires different aspects of performance from foil shapes. All foil shapes should have a broad \( (c_l/c_d) \) vs. \( \alpha \) curve in order to maximize \( C_p \), a well defined but not too abrupt stall, low sensitivity to roughness and minimized minimum pressure peak to mitigate cavitation inception [72].

The root section specifically requires thick foils in order to support the large bending moment associated with the thrust loading. The root also requires a high \( c_{l_{\text{max}}} \). The tip foils should be thin in order to reduce the drag with a high \( (c_l/c_d) \) to maximize performance. The mid-section foils should be a combination of the outboard and root sections.

Once the foils have been selected and assuming ideal twist and taper ratios, a parametric study of blade number, solidity and tip speed ratio can be performed. Figure 1.25 shows the effect of blade number on an optimal rotor design neglecting drag. It is shown that as the blade number increases, the power coefficient increases at a given tip speed ratio [69]. Figure 1.27 shows the effect of drag on a 3 bladed optimum rotor. It is obvious that as the drag increases the power decreases. The effect of solidity without neglecting drag in displayed in Figure 1.26. It can be seen that the addition of drag causes a drop after the peak power is reached as compared to Figure 1.25 which has an asymptotic behavior. As blade number and solidity increases the maximum power increases but once the maximum is reached falls off drastically resulting in a narrow band of maximum power. The lower the solidity the more broad the power coefficient peak is which can
be seen in Figure 1.26. This figure shows that there is an optimum of maximum power
and broad range around 2 or 3 blades, which is why most utility scale wind turbines
are either two or three bladed designs [5]. Adding more blades than three does not add
much of a performance gain and adds weight, cost and structure to the design.

The effect of lift to drag ratio for a three bladed turbine can be seen in Figure 1.27. It
is evident how choosing foils with high lift to drag ratios increases the power production.
Figure 1.25: The effect of blade number on power coefficient of an optimum rotor with zero drag. [69].

Figure 1.26: The effect of solidity on power coefficient. [5].
MHKF1-180s Design Criteria

The foil being tested in this work is the MHKF1-180s. The MHKF1-180s is part of the MHKF1 family of foils seen in Figure 1.28 and was designed to be the foil used in the outboard section of the blade (tip). This family of foils was designed by The University of California Davis to be hydrodynamically and geometrically compatible so they could be utilized on the same rotor blade [72]. The naming convention used for this family of foils can be explained as; “MHKF” is the name of the family and the “1” refers to the year 2011 when they were designed. The numbers on the right side of the dash refer to the percent thickness of the foil multiplied by 10. The “s” at the end of the name refers to anti-singing trailing edge feature. The three foils shown in this figure indicate the root section named the MHKF1-400 (green), the mid-span section named MHKF1-240s (red) and the tip section named MHKF1-180s (blue). When designing this airfoil Shiu et al. [72] formulated the following list of design criteria.

1. High lift to drag ratio in order to maximize efficiency and performance. This is the most basic parameter in the design of a turbine airfoil and is especially important for foils designed for outboard applications. The foil provided was designed to have a maximum lift to drag ratio of approximately 108 based on CFD analysis.

2. Low sensitivity to roughness caused by accumulation of debris, erosion, paint chip-
ping and biological growth among other things. The ability to maintain a sufficiently high lift to drag ratio under a soiled state is a very important design criteria because of the application. The foil was designed to have a delta L/D max of approximately 42 based on CFD analysis.

3. Well defined stall point, but not to abrupt.

4. Low sensitivity to blade surface cavitation. Surface cavitation can erode the surface and cause performance degradation. It is best to design the foil without a large spike at the leading edge pressure peak.

5. The trailing edge of the foil was designed to mitigate trailing edge singing. Trailing edge singing is a phenomenon that commonly occurs on marine propulsors and is undesired due to acoustic constraints in the marine hydrokinetic environment as well as fatigue in the trailing edge.

6. The blade should also be designed with sufficient thickness to withstand the structural loads of underwater operation as well as have enough thickness at the trailing edge to allow for the anti-singing geometry.

1.6.3 Trailing Edge Singing

When designing rotating machinery for use in a marine environment, trailing edge singing may be a concern. Trailing edge singing is when the trailing edge sheds coherent vortex structures that couple with the structural modes and cause the trailing edge to vibrate.
and create an acoustic signature. There are several works that go into great detail about the phenomenon such as Blake [73], Carlton [74], Heskestad and Olberts [75] and Jonson and Eaton [76]. The mechanisms behind trailing edge singing are due to fluid structure interaction. The boundary layers of the pressure and suction surfaces come together at the trailing edge of the foil. If this interaction creates vortex shedding that excites the structural response of the foil this will cause an acoustic signature. This has to be taken into consideration when designing marine turbines because the environmental regulations often have acoustic parameters that must be met. In addition to the noise created from this vortex shedding, structural fatigue is also increased when the trailing edge vibrates.

When a ship propeller sings the trailing edges are often beveled down after installation. In the case of MHK turbines it may be in the designers best interest to design the foil with an anti-singing trailing edge so the performance predictions are more accurate. Anti-singing geometries are asymmetric as to induce separation on the suction side of the foil. The pressure side is designed to remain attached and when they come together the vortices dissipate quickly thus avoiding coherent vortex formation and structural excitation [72].

During the testing procedure which will be described later, two trailing edge designs were tested. The first was a curved trailing edge named the MHKF1-180s and the second was a beveled version of this curved design named MHKF1-180c. When the fin was first fabricated it was done so with the curved trailing edge design and after testing was completed on this model, the trailing edge was beveled down per Figure 1.29. This figure illustrates the trailing edge of the MHKF1-180s from the x/c location of .95 to 1. This figure is the last 95 percent of Figure 1.28 In this figure the hashed part was removed after testing was completed on the original MHKF1-180s foil. After the hashed region of the suction side was removed the foil was given the name MHKF1-180c where the “c” refers to cut. This foil was then tested for performance and trailing edge singing which is addressed in a later section.
1.6.4 Roughness Sensitivity

One of the design criteria for the MHKF1-180s airfoil was low sensitivity to roughness. This was a requirement because the marine environment where the device will be installed has a high probability of roughness occurring. Having a low sensitivity to roughness means that when introduced to roughness, the blade performance does not drop significantly. Roughness on the blade can be anything from algae and barnacle growth to paint chips or dents/dings. Orme et al. [77] performed a study on the effect biofouling had on the efficiency of marine current turbines. In this study barnacle growth was simulated by distributing small cones and testing the foil in a wind tunnel. Several tests were performed using different size roughness elements and different densities of roughness. The results showed a large effect on efficiency. In order to combat biofouling several studies have been performed on foul release coatings and anti-fouling paints [78, 79, 80].

The surface of both of the fabricated foils were finished to 63 micro inches which was determined to be hydrodynamically smooth because this value is within the viscous
sublayer of the boundary layer [81, 82]. The viscous sublayer thickness was found by first calculating the transition point based on an assumed transition Reynolds number of $3 \times 10^6$ on a flat plate [81]. Once the transition point is found the wall shear stress is found using equation 1.17. From this, the wall shear velocity was found using equation 1.18. The viscous sublayer thickness is then found from equation 1.19.

$$\tau = \frac{332 \mu \nu_{\infty}}{x_{tr}} \sqrt{Re_{tr}} \quad (1.17)$$

$$u_\tau = \sqrt{\frac{\tau}{\rho}} \quad (1.18)$$

$$\delta_1 = \frac{5
u}{u_\tau} \quad (1.19)$$

From these equations $\tau$ is the wall shear stress, $\mu$ is the dynamic viscosity, $\nu$ is the kinematic viscosity, $x_{tr}$ is the transition point, $Re_{tr}$ is the transition Reynolds number, $u_\tau$ is the shear velocity and $\delta_1$ is the viscous sublayer thickness. From these equations, the viscous sublayer has a thickness of .0012 inches which is $<$ the surface finish of .000063 inches therefore the foil is hydrodynamically smooth.

In order to test the sensitivity to roughness in a laboratory setting, leading edge distributed roughness is often used. The leading edge distributed roughness prematurely trips the boundary layer from laminar to turbulent which effectively de-cambers the airfoil and reduces the lift curve slope and maximum lift coefficient [83].

Several sources go into details on roughness sizing and the use of leading edge roughness as a transition control mechanism [84, 85, 86, 87]. Freudenreich et al. [87] and van Rooij and Timmer [86] focus on roughness sensitivity for thick wind turbine blade airfoils. Freudenreich specifically focuses on the differences in types of roughness modeling. He examines a trip wire, zig-zag tape and 60 grit carborundum elements to force transition. It was shown that the 60 grit carborundum elements showed the most drastic
reduction in lift and L/D. This is because the boundary layer was immediately tripped at the leading edge and was much thicker than the other methods of transition control. Van Rooij [86] studied the roughness sensitivity and dependence on Reynolds number on several different families of airfoils. The focus Van Rooij was mainly on design criteria and code validation, but the conclusions made were that foils designed for the root had low roughness sensitivity due to the rotational effects therefore the design criteria can be focused on obtaining high lift and structural integrity. It was also concluded in this work that the addition of vortex generators reduced the sensitivity to roughness.

The roughness height in order to force transition is covered in several historical works [85, 84, 88]. Roughness should be sized so that it only trips the boundary layer and does not change the pressure profile. Kuiper [88] states that the size of the roughness should be sized so that the roughness Reynolds number exceeds 120 in order to effectively trip the boundary layer. This number was found through experimentation. Roughness Reynolds number is expressed in equation 1.20.

\[ Re_k = \frac{v_k k}{\nu} \quad (1.20) \]

In equation 1.20 \( v_k \), \( k \), and \( \nu \) are the velocity at the top of the roughness height, the roughness height and the kinematic viscosity respectively.

vanDam [89] outlines a work by Braslow and Knox [85] where a simplified method of determining roughness height is presented. The method is based on a non-dimensional roughness height defined in equation 1.21:

\[ \eta_k = \frac{k}{2s} \sqrt{R_s} \quad (1.21) \]

In this equation, \( s \) is the distance from the leading edge to the roughness and \( R_s \) is the Reynolds number based on the distance from the leading edge to the roughness and boundary layer edge conditions which include \( v_k \) and \( \nu \) [89]. It is stated in van Dam’s work that in order for transition to occur a \( Re_k \) value of 600 is needed. It is cautioned
that this value is based off of empirical data and is only an approximation. Figure 1.30 is given from van Dam’s paper and can be used to find the non-dimensional roughness height. From this the roughness height necessary to trip the boundary layer can be calculated from equation 1.21.

![Figure 1.30](image)

Figure 1.30: Non-dimensional roughness height as a function of the ratio of roughness Reynolds number and station Reynolds number. [89]

### 1.6.5 Cavitation

The definition of cavitation according to Holl [90] is, “vapor and gas filled regions created by a localized pressure reduction which is produced by the dynamic action of a fluid in the interim and/or on the boundaries of a liquid state.” In the application of MHK turbines, the low pressure is caused by the acceleration of flow around the foil. For cavitation to occur, the local pressure must drop below the vapor pressure of the operating fluid. Cavitation is a relevant phenomenon in marine turbines because of performance degradation, surface erosion and noise. When the cavity bubbles encounter a region of pressure higher than the vapor pressure of the liquid they collapse. This collapse creates noise and the jets formed by the collapse have potential to pit and erode the surface. Several papers and books exist on general cavitation and bubble dynamics such as a work
by Holl [91] which outlines the nuclei and bubble growth in a fluid, as well as factors that effect cavitation such as air content. Brennen [92] wrote an extensive review on cavitation and bubble dynamics. Another work by Holl [90] is an overview on cavitation as well as an introduction to cavitation scaling. Brandner [93] also has an introductory lecture on cavitation that provides a good overview.

In marine hydro turbines, low pressures are produced primarily by the curvature of the hydrofoil and tip vortices that form due to the tip leakage caused by the pressure difference between the pressure and suction side of the foil. The cavitation number that is used to approximate surface cavitation can be seen in equation 1.22. This cavitation number is related to the minimum pressure coefficient. This means that when the local pressure reaches approximately the negative of the minimum pressure coefficient of the body cavitation inception occurs. This however is only an approximation because other phenomenon such as presence of nuclei, surface effects and bubble dynamics can break the assumption. The cavitation testing presented in this work is two-dimensional in nature so surface cavitation will be the focus.

$$\sigma = \frac{P_{\text{atm}} + \rho gh - P_v}{\frac{1}{2} \rho v_\infty^2} \approx -c_{p\text{min}}$$  \hspace{1cm} (1.22)

**Hysteresis**

Two methods exist in calling cavitation which are desinent and incipient cavitation. Desinent cavitation is called by holding the velocity constant and bringing the foil to a cavitating state by lowering the pressure. The pressure is then increased until all cavitation ceases and this point is then recorded. This form of cavitation is typically more repeatable and is what many laboratories use for determining cavitation performance, such as the experiments performed by Kermeen [94] and Knapp [95] on the NACA 4412 that was used as our baseline data set. Incipient cavitation on the other hand is called by holding the velocity constant at a non-cavitating condition, then slowly reducing the pressure until cavitation bubbles are initially observed. This cavitation
number is sometimes less than the desinent cavitation number and this phenomenon is called cavitation hysteresis. Cavitation hysteresis is outlined in a work by Holl and Treaster [96] and in this work it was concluded that cavitation hysteresis is random in nature and that increasing the velocity, size and air content tends to decrease the delay time.

**Reynolds Number Effect**

Like many other flow phenomena, cavitation is sensitive to Reynolds number. For cambered airfoils as the Reynolds number increases the critical cavitation number increases as shown in a work by Holl and Wislicenus [97]. Reynolds number effect was of little concern in this work because the full scale and model scale Reynolds numbers were similar. The Reynolds number did change in some of the cavitation testing, but it was within a range to have minimal effect on the data.

**Gas Content Effect**

Dissolved gas in the water effects the cavitation performance of a foil. The PSU/ARL 12-inch diameter water tunnel has the ability to control the dissolved gas content by going on a bypass system that removes the air from the solution. The air content is then measured using a Van Slyke apparatus. Dissolved gas effects cavitation inception in two ways; the first being the free gas acts as a cavitation nuclei and second the dissolved gas influences the size of the nuclei and the growth of cavitation bubbles [91]. Holl [91] summarized work performed at the University of Minnesota where a test was conducted to study the effect of air content on the cavitation inception of a 1.5 caliber ogive. The result was that as the air content increased, the cavitation inception number increased for a given velocity as seen in Figure 1.31.

The gas content of the liquid also governs whether the cavitation is gaseous or vaporous. Gaseous cavitation occurs when a liquid is over saturated therefore is not a concern in this study. All cavitation mentioned in this thesis is vaporous cavitation
which is when the liquid rapidly converts from liquid to vapor [98].

A work by Naylor and Millward [99] summarizes a method of predicting the effect of dissolved gas on cavitation inception of a tip vortex. The results of this work showed that the prediction method shows good agreement with experimental data and that air content has a significant effect on inception at speeds lower than 4 m/s.

Roughness Sensitivity

Like previously mentioned, the sensitivity of roughness is an important parameter in this particular foil design. Roughness not only effects the forces, it also effects the cavitation performance. The addition of roughness causes cavitation earlier than when the foil is
in a clean condition. The roughness sensitivity conducted in this work was broken down into two types of roughness which were distributed roughness and isolated roughness. The distributed roughness was 60 grit (254 micron average) carborundum distributed over the first 7 percent of the chord so that the roughness elements covered 50 percent of the area. The isolated roughness was 46 (356 micron average) and 16 (1092 micron average) grit carborundum elements strategically placed at different locations along the chord.

A Work by Arndt et al. [100] investigates the effects of both distributed and isolated roughness elements and summarizes different forms of roughness that may form. In this work it is noted that cavitation on isolated roughness elements can be of two forms depending on the roughness shape. Triangular roughness elements cavitate in the shear layer downstream of the point of cavitation while rounded streamline protrusions cavitate on the surface of the protrusion [100]. In both cases the critical cavitation number increases as the height of the roughness increases. In the same work, distributed roughness was analyzed and the conclusion is that distributed roughness is more complex than isolated roughness because of the interrelationship of the boundary layer and roughness elements on the surface. Isolated roughness also only produces changes in the flow local to the roughness element and does not change the large scale flow. The cavitation performance is changed by distributed roughness because cavitation forms in the large eddies in the boundary layer due to the increased turbulence [100]. The conclusions made in this work were that, for a given roughness size, isolated roughness has a larger effect on cavitation performance than distributed roughness. Also the placement of the isolated roughness was important because it had to be near the minimum pressure point to have the largest effect. It was also concluded that isolated roughness was more Reynolds number dependent than distributed roughness which was said to usually be Reynolds number independent [100].

In ITTC Report of the cavitation committee [101] an overview of isolated and distributed roughness was given. The report states that isolated roughness cavitation in-
ception is dependent on the turbulence intensity, Reynolds number, roughness height and presence of suitable nuclei [101]. The report also states that distributed roughness is independent of nuclei content because the strong eddies and separation will generate nuclei [101].

**Cavitation Breakdown**

The influence cavitation has on the forces of a NACA 66 (MOD) hydrofoil are examined in a work by Shen and Dimotakis [102]. In this work cavitation was classified as leading edge sheet cavitation and mid chord cavitation. The former is when cavitation forms off the leading edge and the latter is when cavitation occurs mid chord of the foil. It was found in this work that both of these forms of cavitation increased the lift of the foil when the cavity does not extend beyond 83 percent of the chord. The increase in lift due to mid chord cavitation was much smaller than leading edge sheet cavitation. The authors found that the increase in lift due to the cavitation was cause by a lower than normal pressure beneath the cavity due to the vaporization of water. Mid chord cavitation had a smaller effect on the lift because the leading edge pressure peak was smaller due to a thickness effect caused by the cavitation. Once the cavity progressed passed 83 percent chord there was a large drop off in lift due to separation. For both forms of cavitation drag increased due to increased form drag from the cavitation [102]. For some foils, this is not the case such as those designed for super cavitating conditions which are not considered in this work. Figure 1.32a from this work shows the increase in lift and drag as a function of cavity length while Figure 1.32b shows the lift coefficient as a function of cavitation number.

1.7 **NACA 4412 Historical Data**

The NACA 4412 airfoil was chosen to validate the test because of the amount of historical wind and water tunnel data publicly available. The existing wind tunnel data comes from
tests performed in the NACA variable density wind tunnel [103, 104, 105, 82], Langley two-dimensional low turbulence wind tunnel [106] and Texas A&M wind tunnel [107]. Previous water tunnel tests were conducted at Cal. Tech [94, 95]. The historical data was compared and can be seen in Figures 1.33 and 1.34. The lift data is within good agreement in the linear range of the lift curve slope with the exception of the Abbott [82] data. It is only when the flow begins to separate and the foil begins to stall that the results begin to diverge and this divergence is still relatively small. The Abbott [82] data does not agree as well with the other data sets because it is at a higher Reynolds number of 3 million. The data from Abbott [82] was taken at 3, 6 and 9 million Reynolds number and shows an increase in maximum lift and overall decrease in drag with increasing Reynolds number. The drag data is in good agreement at the angles of attack where the flow remains attached but the differences in the data sets where separation begins to occur is much more apparent in the drag data. This variation in the data may be due to differences in testing methods, tunnel setup, turbulence intensity, model surface finish and load measurement capability or a combination. Fuglsang and Bove [108] came to the conclusion that there is a high uncertainty when comparing results from different
tests in different tunnels.

Figure 1.33: NACA 4412 historical lift data at comparable Reynolds numbers.[82, 94, 95, 104, 106, 107]

Figure 1.34: NACA 4412 historical drag data at comparable Reynolds numbers.[82, 94, 95, 104, 106, 107]

The Cal. Tech tunnel used by Kermeen in 1956 [94] was a 14 inch diameter closed jet water tunnel. In order to simulate two dimensional flow, an insert was created for the tunnel 14 inches high and 3 inches wide creating a channel. The fin chord was 3 inches
which resulted in an aspect ratio of 1. It is claimed that the angle of attack is accurate to 1 minute of arc. The fin was installed so that there was no gap at the base where the fin was attached to the force balance and the gap at the tip was held to within .002 in. Tunnel blockage corrections were applied to the data based on the same methods presented in Barlow et al. [109].

Knapp and Daily performed a test in the same tunnel at Cal. Tech before Kermeen in 1944 [95]. In this test the model fin had a chord of 3 inches and an aspect ratio of 3.3. Gaps were held at approximately .005 inches. No corrections were applied to this data.

The NACA variable density tunnel which was used for the test by Stack, Abbott and Jacobs [103, 82, 104] features an outer pressure vessel that contains the tunnel inside. The outer pressure vessel can withstand pressures up to 21 atmospheres which changes the operating Reynolds number by increasing or decreasing the air density via tank pressurization [110]. Turbulence is mitigated in this tunnel through anti-swirl vanes and screens downstream of the fan but despite of this, the tunnel has a relatively high turbulence intensity of 2% [111]. The tests run in this tunnel were three dimensional tests where the data was then corrected to two dimensional. This could be a source of some error and discrepancies in the data between tunnels. A schematic of the NASA variable density tunnel is shown in Figure 1.35.

The Langley two-dimensional low turbulence wind tunnel is a closed loop wind tunnel design as seen in Figure 1.35. The Langley two-dimensional low turbulence tunnel was designed to operate only at atmospheric pressure and Reynolds number is varied by increasing or decreasing the tunnel velocity assuming a given model chord. The turbulence was mitigated through seven screens placed upstream of the contraction section. Free stream turbulence intensities vary with free stream speed and range from a few hundredths of a percent at the lowest speeds to tenths of a percent at the higher speeds [112]. The Reynolds number dependence of turbulence intensity for this tunnel can be seen in Figure 1.37.

The Texas A&M wind tunnel use by Ostowari in 1984 [107] is a closed circuit wind
Figure 1.35: Schematic of the NACA variable density wind tunnel[110].

Figure 1.36: Schematic of the NACA low turbulence wind tunnel[112].

tunnel with a rectangular test section measuring 2.13 meters in height and 3.05 meters in width. The turbulence intensity was published as less than 1% [113]. The test was an investigation of the post stall characteristics of several NACA 44XX sections of varying thickness including the NACA 4412. In this test the angle of attack of the foils ranged from -10 to 110 degrees. The data was taken at several different Reynolds numbers
and several different aspect ratios, and the data used in this work utilized the 1 million Reynolds number and infinite aspect ratio. Only the lift data was used from this study because the drag results could not be used due to the resolution of the plots.

Figure 1.38: The Texas A&M Wind Tunnel.[113].
Experimental Setup

2.1 Applied Research Lab 12-Inch Diameter Water Tunnel Facility

The PSU/ARL 12-inch diameter water tunnel is a closed circuit, closed jet water tunnel with two 0.762 m (2.5 ft) long tests sections; an axisymmetric test section with a 0.3048 m (1 ft) diameter, and a 114mm by 508mm (4.49 in by 20 in) rectangular test section [114]. A variable speed pump produces test section speeds up to 21 m/s (68.9 ft/s). Turbulence control is through a 152 mm (5.98 in) deep section of honeycomb with a 25 mm (.98 in) core size installed in the upstream plenum of a 9:1 (11.3:1 rectangular test section) area contraction ratio inlet nozzle. The test section turbulence intensity is no more than 0.3% over the velocity range [115]. Test section static pressure can be adjusted between 14 kPa and 414 kPa (2.03 psi and 60.04 psi) absolute over the full velocity range. Water quality, which affects cavitation performance, is controlled by a bypass system designed to remove dissolved gas and particulates. Tunnel operating conditions including free stream velocity, water temperature and static pressure, are continuously monitored throughout a test. Free stream velocity is measured using a kiel probe and static tap to measure total and static pressure respectively, a Pitot-static probe mounted in the test section or Laser Doppler Velocimetry (LDV). The Pitot-static probe and kiel probe/static tap are
used for continuous velocity monitoring. Water temperature and pressure are monitored with StoLab RTD probes and Heise piezoelectric absolute pressure gauges, respectively. The test sections have optical access for flow diagnostics as illustrated in Figure 2.1.

![Figure 2.1: 12-inch diameter water tunnel schematic illustrating both circular and rectangular test sections.](image)

**2.2 Test Hardware**

Several unique pieces of hardware were created for these tests. Figure 2.2 illustrates the view of the test setup from the bottom of the test section. An existing window was modified to accommodate the fin setup for this test. Tick marks were added to the window every one degree in order to measure the angle of attack. At every degree marker there is a pin hole and an arm which locks into these pin holes in order to insure repeatability of angle of attack. A new load cell can and base were created. The base
mounted flush to the bottom window and allowed the angle of attack to be changed using slotted bolt holes. Figure 2.2 illustrates the view of the test setup from the bottom of the test section. A large handle, which is not shown in Figure 2.2, was attached to the base and was used to rotate the fin while the tunnel was filled and pressurized. The locking pin was then inserted into the proper angle hole and the bolts on the base were tightened down to lock in the angle. The load cell was mounted to the inside of the load cell can using 4 pin holes on the bottom of the can in which the load cell pins are aligned to insure no rotation and accurate alignment. Pins were also placed on the top surface of the load cell which aligned with pin holes in a mounting post seen in 2.3b. On the top surface of the mounting post were two more pins which aligned with holes in the fin to insure no rotation.

![Figure 2.2: PSU/ARL 12-inch diameter water tunnel window modifications.](image)

2.2.1 **NACA 4412 Solid Fin**

Two versions of the NACA 4412 fin were fabricated for this test. The first fin installed in the tunnel was a solid fin that spanned the entire width of the test section. The span,
chord and thickness of the fin were 113.54 mm (4.47 in), 203.2 mm (8 in) and 24.4 mm (.96 in) respectively. A cross section of this full span fin can be seen in Figure 2.3a. When the fin was installed the end gaps at the base and tip were less than .5 mm (.019 in). The fin installed in the test section can be seen in Figure 2.4. In order to minimize the gap size, appropriate sized shim stock was attached to the foil. The stains on the fin seen in the picture are due to an Alodine treatment and did not affect the surface of the fin.

![Figure 2.3: Solid foil and mounting post two-dimensional cross sections. Dimensions are in inches.](image)

**2.2.2 NACA 4412 and MHKF1-180s Three Part Fins**

In addition to the NACA 4412 solid fin, a three part fin was also fabricated for both the NACA 4412 and MHKF1-180s fins. A solid fin was not fabricated for the MHKF1-180s fin because the data for the two NACA 4412 fins were in good agreement. The three part fin was a unique design in which only the middle section was instrumented by the load cell and was separated from the end sections in order to reduce three-dimensional effects that may bias two-dimensional flow, such as end wall effects and gap flow. An isometric CAD drawing of the three part test setup can be seen in Figure 2.5.
Figure 2.4: NACA 4412 solid fin installed in the test section showing gaps.

Figure 2.5: NACA 4412 three part fin CAD drawing.
The middle section of the foil instrumented by the load cell (shown in light blue) can be seen attached to the mounting post (shown in yellow). The two end pieces are separated from the middle section by two spacer rods (seen in brown). The two outer sections are isolated from the center foil section (seen in purple). The bottom section is directly mounted to the window mounting base (shown in dark green). The top section is mounted to the lower section through spacer rods (seen in brown) which pass cleanly through the center section and bolt to the lower foil section joining the outer and inner pieces allowing the middle piece to "float" while supported only by the spacer rod which is attached directly to the load cell (shown in pink). The gaps between the instrumented section and the end sections were controlled using shim stock and maintained at approximately .254 mm (.010 in). An exploded view of the entire window test setup can be seen in Figure 2.6.

2.3 Measurement Instrumentation

Tunnel free stream velocity was measured by both a Pitot-static probe and tunnel wall static and total pressure measurements to provide a redundant velocity measurement. Total pressure was measured by a Kiel probe mounted in the settling chamber upstream of the test section contraction nozzle. The pressures were measured using Heise pressure transducers [116]. Each pressure transducer has an accuracy of ±0.02% full scale output.

The temperature of the water was measured using a STOLAB temperature sensor [117]. The accuracy of the sensor is ±.10° C (±.15° F).

2.3.1 AMTI Six Degree of Freedom Load Cell

The forces were measured using a water proof AMTI Inc. six degree of freedom load cell (SP2.5D-1K-6010). The load cell specifications and capacities/sensitivites can be seen in Table 2.1 and 2.2 respectively. The load cell signals were amplified using an AMTI mini amp (MSA-6) and digitized using a national instruments data acquisition system.
Figure 2.6: Exploded view of the 3 part test assembly.
(NI USB-6295) with an in house generated LabVIEW code.

Table 2.1: AMTI SP2.5D-1K-6010 load cell specifications

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensions (length x diameter)</td>
<td>2.5 x 2.5 in</td>
</tr>
<tr>
<td>Weight</td>
<td>1 lb.</td>
</tr>
<tr>
<td>Channels</td>
<td>Fx, Fy, Fz, Mx, My, Mz</td>
</tr>
<tr>
<td>Top plate material</td>
<td>Aluminum</td>
</tr>
<tr>
<td>Temperature range</td>
<td>0 to 125°F</td>
</tr>
<tr>
<td>Excitation</td>
<td>10V maximum</td>
</tr>
<tr>
<td>Sensing elements</td>
<td>Strain gage bridge</td>
</tr>
<tr>
<td>Amplifier</td>
<td>Required</td>
</tr>
<tr>
<td>Analog outputs</td>
<td>6 Channels</td>
</tr>
<tr>
<td>Digital outputs</td>
<td>None</td>
</tr>
<tr>
<td>Crosstalk</td>
<td>&lt;2% on all channels</td>
</tr>
<tr>
<td>Fx, Fy, Fz non-linearity</td>
<td>± 0.2% full scale output</td>
</tr>
<tr>
<td>Fx, Fy, Fz hysteresis</td>
<td>± 0.2% full scale output</td>
</tr>
</tbody>
</table>

Table 2.2: AMTI SP2.5D-1K-6010 load cell capacities and sensitivities

<table>
<thead>
<tr>
<th>Channel</th>
<th>Fx</th>
<th>Fy</th>
<th>Fz</th>
<th>Units</th>
<th>Mx</th>
<th>My</th>
<th>Mz</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Capacity</td>
<td>500</td>
<td>500</td>
<td>1000</td>
<td>lb</td>
<td>1000</td>
<td>1000</td>
<td>500</td>
<td>in-lb</td>
</tr>
<tr>
<td>Sensitivity</td>
<td>2.4</td>
<td>2.4</td>
<td>0.6</td>
<td>v/v-lb</td>
<td>3</td>
<td>3</td>
<td>2.4</td>
<td>v/v-in-lb</td>
</tr>
<tr>
<td>Natural frequency</td>
<td>-</td>
<td>-</td>
<td>0</td>
<td>Hz</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>Hz</td>
</tr>
<tr>
<td>Stiffness x10^5</td>
<td>1</td>
<td>1</td>
<td>17</td>
<td>lb/in</td>
<td>-</td>
<td>-</td>
<td>2</td>
<td>in-lb/rad</td>
</tr>
</tbody>
</table>

Due to the size of the model which was tested, the moment measured by the load cell approached the upper limit of the load cell capacity at the test chord Reynolds number of 1.3 million. This limited the range of Reynolds numbers where forces could be measured. It was necessary to increase the tunnel speed for several conditions in order to study the cavitation performance. To avoid damage to the load cell, an aluminum “dummy” load cell was built.

2.3.2 Laser Doppler Velocimetry

Laser Doppler Velocimetry (LDV) profiles of the foil wake flow were taken at several angles of attack in order to compare with the load cell data. A TSI Inc. fiber-optic, multi-component, laser Doppler velocimeter was used for the velocity measurements in
the 2-D foil tests in the twelve inch tunnel’s rectangular test section. The output of a Coherent Innova-70 5 Watt Argon-Ion laser is transmitted through a TSI Inc. color separator frequency shift unit (Fiber-Light Model FBL-3). The Fiber-Light splits the incoming multi-mode laser beam exiting the laser into two beams of equal intensity and then separates each beam into three single-mode wavelengths of light (476.5, 488 and 514.5 nm). A Bragg cell incorporated into the beam path within the fiber-light adds a 40 MHz frequency shift to one beam in each single-mode beam pair. The 514.5 nm and 488 nm wavelength beam pairs are coupled to a TSI 83 mm, two-component fiber-optic transceiver probe (Model 9832). A 245 or 363 mm focal length lens was used to transmit and focus the laser beams into the tunnel. The measurement volume characteristics for this optical setup in water can be seen in Table 2.3.

Table 2.3: LDV laser probe volume properties.

<table>
<thead>
<tr>
<th>Beam diameter (mm)</th>
<th>Focal Length (mm)</th>
<th>Beam spacing (mm)</th>
<th>kappa</th>
<th>Probe length (mm)</th>
<th>Probe diameter (mm)</th>
<th>Fringe spacing (microns)</th>
<th># of fringes (-)</th>
<th>Wave length (nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.82</td>
<td>245</td>
<td>50</td>
<td>4.38</td>
<td>0.74</td>
<td>0.06</td>
<td>2.53</td>
<td>23</td>
<td>514.5</td>
</tr>
<tr>
<td>2.82</td>
<td>245</td>
<td>50</td>
<td>4.38</td>
<td>0.71</td>
<td>0.05</td>
<td>2.4</td>
<td>23</td>
<td>488</td>
</tr>
<tr>
<td>2.82</td>
<td>363</td>
<td>50</td>
<td>2.66</td>
<td>1.81</td>
<td>0.08</td>
<td>3.74</td>
<td>23</td>
<td>514.5</td>
</tr>
<tr>
<td>2.82</td>
<td>363</td>
<td>50</td>
<td>2.66</td>
<td>1.72</td>
<td>0.08</td>
<td>3.55</td>
<td>23</td>
<td>488</td>
</tr>
</tbody>
</table>

The LDV system was operated in the backscatter mode collecting scattered light from 10 micron diameter hollow glass micro spheres (Potters Industries - Sphericel) used as seed. The scattered light is received by the transceiver probe through the transmitting lens system, and fiber-optically transmitted to a TSI photo-multiplier system (PDM 1000). The PDM 1000 converts the scattered light into a frequency modulated analog signal at a frequency of oscillation proportional to the speed of a seed particle traversing the measurement volume and the Bragg cell frequency shift. This signal is then processed using a TSI FSA3500 Doppler signal processor to calculate the signal frequency, correct for downmixing and convert the measured frequency to the particle velocity. The axial component was downmixed at a frequency of 2 MHz to eliminate directional ambiguity.
and reduce fringe bias. Flowsizer software (TSI Inc.) was used to control the FSA3500 processor and acquire velocity data. In-house developed post processing software was used to further process the acquired LDV data to compute velocity statistics, perform noise filtering and apply velocity bias corrections. The LDV system is mounted on a computer controlled 3-axis traverse table which automatically positions the probe to collect data at several predetermined points across the span of the tunnel. The traverse table uses stepper motor control to provide linear actuator positioning control as well as linear encoders on each axis to provide position feedback. This traverse table was synced with the LDV capture software in order to automate the LDV survey runs. The laser setup is illustrated in Figure 2.7. The LDV capture software used was Flowsizer by TSI [118]. This software acquired the data as well as controlled the traverse table.

![Figure 2.7: LDV hardware setup.](image)

### 2.3.3 Flow Visualization

**Oil-Paint Surface Flow Visualization**

In order to qualitatively assess the flow field on the foil, Oil-Color artist paint was mixed with 80w-90 gear oil and added to the blade surface in small dots or blobs. Zierke et. al. [119] describes the type of flow that can be visualized with surface paint flow visualization as well as the proper mixing and application techniques and Table 2.4 shows the mixing ratio based on test speed. The consistency of the mixture for this test
was that of yogurt.

Table 2.4: Oil-Paint to gear oil mix ratios based on testing speed.

<table>
<thead>
<tr>
<th>Expected Relative Velocity (ft/sec)</th>
<th>Percentage of Gear Oil in the Oil-Paint Mixture</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>70%</td>
</tr>
<tr>
<td>40</td>
<td>50%</td>
</tr>
<tr>
<td>60</td>
<td>30%</td>
</tr>
<tr>
<td>80</td>
<td>10%</td>
</tr>
</tbody>
</table>

Adding two different colors proved to be beneficial in the event there was any mixing of the flow by complex flow features. Paint was also added to the baseplate for some of the runs to attempt to capture the end wall junction vortex. Examples of the paint application can be seen in Figure 2.8.

**Yarn Tuft Flow Visualization**

Yarn tufts were also applied to the surface of the fin to show the real time flow over the foil. Small pieces of yarn approximately 6.35 mm (.25 in) in length were super glued to the blade surface spaced evenly over the chord and span. Figure 2.8b illustrates the tuft application.

(a) Oil-Paint flow visualization paint application using two different colors prior to run.  
(b) Application of yarn tufts to the blade surface for flow visualization.

Figure 2.8: Flow visualization techniques.


2.4 Cavitation Calls

Cavitation was called on all tests visually by the author to reduce uncertainty in the call. A strobe light set to a frequency of 120 Hz was used to aid in the visualization of the cavitation. A Casio Exilim 9.1 megapixel camera was used to photograph and video record the cavitation. Inception and desinence cavitation was called for several of the tests in order to show any hysteresis effects, but the data presented and compared to the historical data is desinence unless otherwise noted. Hysteresis effects will be addressed in a later section. Cavitation was called on the blade only, and all cavitation due to the gaps were neglected in the cavitation call and often disappeared before blade cavitation disappeared. Inception critical cavitation number was called by lowering the pressure until the first bubble appeared on the foil cavitation then taking data at that point. Desinence critical cavitation number was called by increasing the pressure and when the last cavitation bubble disappeared and did not return the data was recorded. During the tests involving roughness, cavitation was called when cavitation disappeared on the roughness elements. Air content was monitored during all cavitation tests. Figure 2.9 shows cavitation very close to the point of inception for both the clean NACA 4412 and MHKF1-180s with distributed roughness.

2.5 Addition of Roughness

Several cases were run on both the NACA 4412 and MHKF1-180s foils where roughness in the form of carborundum elements was applied to the foil using spar urethane as an adhesive. Spar urethane was used since it was able to adhere to the surface during testing without breaking due to water and also allowed removal of the roughness minimizing permanent surface damage.

In order to artificially transition the boundary layer leading edge, distributed roughness was added in order to represent real world events such as deposition of sediment on the leading edge, formation of algae, widespread biofouling or many other forms of
debris common to a marine environment. Orme et al. [77] investigated the effect of biofouling on the efficiency of marine current turbines and showed a large effect therefore illustrating the importance of understanding this phenomenon [77]. Distributed roughness was simulated by applying 60 grit carborundum elements over the first 7% of the chord so that the roughness covered roughly 50% of the area. The roughness was applied using spar urethane and blowing particles onto the painted area. This method was based on previous wind tunnel tests on roughness effects on wind turbine airfoils [87] as well as tests done showing cavitation performance under the influence of surface irregularities [100]. The application of the distributed roughness can be seen on the suction side of the MHKF1-180s in Figure 2.11a. This case represented an extreme scenario where the boundary layer was tripped very close to the leading edge.

Another test case involved installing a trip wire at the 7% chord location. The wire was .016 inches in diameter and was applied with super glue. This case was different from the distributed roughness because the boundary layer was allowed to develop up to the 7% location, then was tripped to turbulent. In cases where the angle of attack was greater, the boundary layer may have already been turbulent when the trip wire was
encountered.

In the final roughness case, isolated roughness was simulated using 46 and 16 grit carborundum elements strategically placed along the chord. The elements were staggered along the chord so that a roughness element could coincide with the minimum pressure location as angle of attack changed. The locations of the minimum pressure were estimated using XFOIL [120] before applying the roughness. The locations of the isolated roughness can be seen in Figure 2.10. The individual elements were spaced spanwise in a way where interference on a downstream element was minimized. The elements were applied with a small amount of spar urethane. The locations of the isolated roughness elements can be seen in Figure 2.11b. Cavitation was called on the isolated roughness foil by establishing developed cavitation on the foil and then increasing the pressure until cavitation disappeared on the last roughness element.

![Figure 2.10: Locations of isolated roughness elements based on XFOIL analysis of minimum pressure location.](image)

### 2.6 Anti-Singing Trailing Edge Spectral Analysis

LDV trailing edge spectral analysis was conducted on two anti-singing trailing edge designs as well as the baseline NACA 4412 foil. The data were acquired using the same LDV system as listed above. The LDV probe volume was positioned 5 mm down stream of the trailing edge of the foil being tested. A small LDV velocity survey was then taken
approximately 5mm above and below the trailing edge in order to find the location of maximum velocity RMS. The velocity profile grid can be seen in Figure 2.12. Once the point of maximum RMS was found, the LDV probe volume was placed at this location and 1 million points were collected at the speeds of 5, 6, 8 and 10 mps. The data rate for these tests varied, but was on average 3500 Hz. Once the velocity data was taken at these speeds, it was processed using in house codes to produce a spectra which is presented later in this work.

Figure 2.12: LDV trailing edge survey grid used to find the location of maximum RMS. Red dots indicate locations velocity was measured.
Data Reduction and Corrections

3.1 Data Reduction Equations

Load Cell Coordinate System Transform

A coordinate transform was necessary due to the rotation of the load cell as the fin changed angle of attack. The raw data in the load cell coordinate system \((F_x, F_y)\) is rotated into the tunnel coordinate system \((\text{Lift}, \text{Drag})\). The coordinate system used in the test can be seen in the following diagram illustrated in Figure 3.1. The variables \(F_x\) and \(F_y\) refer to the force in the x and y directions of the load cell respectively and \(\alpha\) is the fin angle of attack.

In order to obtain lift and drag from the load cell coordinate system, equations 3.1 and 3.2 were used respectively.

\[
L = F_y \cos(\alpha) + F_x \sin(\alpha) \tag{3.1}
\]

\[
D = F_y \sin(\alpha) - F_x \cos(\alpha) \tag{3.2}
\]

In the case of the NACA 4412, the mounting point of the load cell was at the quarter chord point, but did not lie on the chord line of the foil therefore affecting moment
calculation and requiring an additional offset transform. The MHKF1-180s foil was mounted at the quarter chord point on the chord line so no transform was necessary. When the NACA 4412 was at zero degrees angle of attack, the load cell was offset 6.35 mm (.25 in) above the chord line and is represented by the variable \( l \) in Figure 3.1. When rotated, this resulted in an offset in the x and y directions indicated by equations 3.4 and 3.3 respectively.

\[
y = l\cos(\alpha) \tag{3.3}
\]

\[
x = l\sin(\alpha) \tag{3.4}
\]

The moment contribution due to the offset in the lift and drag directions are represented by equations 3.5 and 3.6. To find the total moment about the quarter chord the lift and drag moments were added to the moment recorded by the load cell which is represented by the variable \( M_z \) and seen in equation 3.7.
\[ M_{Lift} = F_y \cos(\alpha) y - F_y \sin(\alpha) x \]  
\[ (3.5) \]

\[ M_{Drag} = -F_x \cos(\alpha) y - F_x \sin(\alpha) x \]  
\[ (3.6) \]

\[ M = M_{Lift} + M_{Drag} + M_z \]  
\[ (3.7) \]

**Force Coefficients**

The normalized force coefficients were calculated in order to compare to historical data. The force data transformed from the load cell is not corrected for tunnel blockage at this point and the uncorrected lift, drag and quarter chord moment coefficients are represented by equations 3.8, 3.9 and 3.10 respectively.

\[ c_{lu} = \frac{L}{\frac{1}{2} \rho v_\infty^2 S} \]  
\[ (3.8) \]

\[ c_{dvu} = \frac{D}{\frac{1}{2} \rho v_\infty^2 S} \]  
\[ (3.9) \]

\[ c_{ml/4u} = \frac{M}{\frac{1}{2} \rho v_\infty^2 S c} \]  
\[ (3.10) \]

The force data is normalized by the water density \((\rho)\), velocity \((v_\infty)\) and fin area \((S)\). The moment data has an extra term in the normalization which is the chord of the fin \((c)\). The fin area is defined as the fin chord multiplied by the span. It is worth noting that the three part fin was normalized by the area of the instrumented section of the foil.

The cavitation number is used to quantify the conditions where cavitation occurs as seen in equation 3.11. Cavitation is quantified by subtracting the local pressure from the vapor pressure of the liquid then normalizing by the local dynamic pressure. The
cavitation number is a function of depth due to the increased local pressure, but in tunnel tests the pressure is controlled so the local pressure is the tunnel pressure, so in equation 3.11 $P_{atm} + \rho gh$ is replaced with $P_{tunnel}$.

$$\sigma = \frac{P_{atm} + \rho gh - P_v}{\frac{1}{2} \rho v_\infty^2}$$

(3.11)

### 3.2 Gap Corrections Applied to Load Data

The first correction that was done to the data after the coordinate transform was to account for the gap effects. Due to the test hardware, gaps existed that needed to be corrected for. These gaps introduced unwanted three-dimensional effects that biased the flow and must be subtracted out. The first gap corrections made were on the solid NACA 4412. As stated in section 2.2.1 the gaps on the solid NACA 4412 fin were approximately .381 mm (.020”). A hydrofoil end gap experiment was performed by Kermeen [94]. This experiment was conducted on a NACA 4412 hydrofoil and experimented with the effect of different sized gaps on forces. The effect of lift, drag and moment as a function of gap for 0 and 4 degrees angle of attack can be seen in Figure 3.2 produced from Kermeen [94].

![Figure 3.2: Gap effect on the force coefficients for two angles of attack][94]
The data was extracted from this plot for each gap size for both 0 and 4 degrees and extrapolated into a linear curve to obtain the gap effect as a function of angle of attack. From Figure 3.2 it can be seen that there is negligible effect of gap on the moment coefficient, therefore no correction was applied to this coefficient. The change in $c_l$ and $c_d$ as a function of angle of attack can be seen in Figures 3.3 and 3.4 respectively. In these figures, the $y$ variable represents delta $c_l$ while the $x$ variable represents the angle of attack ($\alpha$). In order to correct the experimental results, the equations for .020" were applied to the data by adding the delta $c_l$ and subtracting the delta $c_d$. These equations are represented by equations 3.12 and 3.13. It can be seen that lift is decreased and drag is increased due to the effect of the gap.

![Figure 3.3: Change in $c_l$ for different gap sizes as a function of angle of attack.](image)

\[
\Delta c_l = .0022\alpha + .0187 \tag{3.12}
\]

\[
\Delta c_d = -.0009\alpha - .0019 \tag{3.13}
\]
3.3 Tunnel Blockage Corrections

The interference effects of the tunnel walls cause extra forces on the model that must be corrected. The differences between a tunnel test and operation in an open environment are called blockage effects. These effects are corrected using methods found in Barlow et. al. [109]. The solid blockage due to the presence of the fin in the tunnel and the wake blockage due to the tunnel restraining the wake expansion were the effects accounted for in the experiment. The effect due to buoyancy caused by a longitudinal static pressure gradient was not included because the ARL 12 inch tunnel was designed to have a zero longitudinal static pressure gradient. The blockage is corrected by calculating factors $\epsilon_{sb}$ and $\epsilon_{wb}$ which are the solid and wake blockage factors respectively illustrated in equations 3.14 and 3.15. The blockage factor, $\epsilon$, is the summation of these two factors as seen in equation 3.16

$$\epsilon_{sb} = 0.822 \lambda^2 \frac{t^2}{h_t^2}$$ (3.14)

$$\epsilon_{wb} = \frac{c}{h_t} \frac{1}{4} c_{du}$$ (3.15)
\[ \epsilon = \epsilon_{sb} + \epsilon_{wb} \]  

(3.16)

In these equations \( t, c, h_t \) and \( c_{du} \) refer to airfoil thickness, chord, test section height and uncorrected drag coefficient respectively. The term \( \lambda_2 \) is a factor found in Barlow et. al. [109] that approximates an airfoil shape from a cylinder and is plotted in figure 3.5. The blockage correction equations are found in Table 3.1.

![Figure 3.5: Lambda 2 Factor for different streamline shapes. Fineness ratio refers to chord/thickness [109].](image)

Table 3.1: Barlow et. al. blockage correction equations [109].

<table>
<thead>
<tr>
<th>Equation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_{Block} = \frac{v^2}{2s} \left( \frac{c}{h_t} \right)^2 )</td>
<td>Sigma Factor</td>
</tr>
<tr>
<td>( v_{\infty} = v_u (1 + \epsilon) )</td>
<td>Velocity Correction</td>
</tr>
<tr>
<td>( q = q_u (1 + 2\epsilon) )</td>
<td>Dynamic Pressure Correction</td>
</tr>
<tr>
<td>( Re = Re_u (1 + \epsilon) )</td>
<td>Reynolds Number Correction</td>
</tr>
<tr>
<td>( \alpha = \alpha_u + \frac{57.3\sigma_{Block}}{2\pi} (c_{lu} + 4c_{m1/4u}) )</td>
<td>Angle of Attack Correction in Degrees</td>
</tr>
<tr>
<td>( c_l = c_{lu} (1 - \sigma_{Block} - 2\epsilon) )</td>
<td>Lift Coefficient Correction</td>
</tr>
<tr>
<td>( c_{d0} = c_{d0u} (1 - 3\epsilon_{sb} - 2\epsilon_{wb}) )</td>
<td>Drag Coefficient Correction</td>
</tr>
<tr>
<td>( c_{m1/4} = c_{m1/4u} (1 - 2\epsilon) + \frac{\sigma_{Block}c_l}{4} )</td>
<td>Moment Coefficient Correction</td>
</tr>
</tbody>
</table>


3.4 Error Analysis

An error analysis was conducted on all experimental data sets using the methods presented in Coleman and Steele [121]. The sources of measurement error in this experiment came from the load cell, pressure transducers and temperature probe. Along with the resolution error in the load cell, repeatability, hysteresis and zero drift error were also accounted for. The random error was accounted for using equation 3.17.

\[
\delta_{\text{rand}} = \frac{1.96 \sigma_{\text{std}}}{\sqrt{N}} \tag{3.17}
\]

The error values for the load cell were given by the manufacturer specifications and can be seen in Table 3.2. The z component is included for completeness but was not used due to the two-dimensional nature of this experiment. The zero drift error was also included in the force error analysis on a case by case basis because this error varied from run to run. The pressure transducers and temperature probe errors were given by the manufacturers as well and are ±.02 psi [116] and ±.15° F [117] respectively which corresponds to an error in density measurement of .004 slugs/ft³.

<table>
<thead>
<tr>
<th>Error Source</th>
<th>FX (lbs.)</th>
<th>FY (lbs.)</th>
<th>FZ (lbs.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resolution</td>
<td>0.617</td>
<td>1.56</td>
<td>1.827</td>
</tr>
<tr>
<td>hysteresis</td>
<td>1</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>non-linearity</td>
<td>1</td>
<td>1</td>
<td>2</td>
</tr>
</tbody>
</table>

With all the measurement errors known, the errors were propagated through the data processing equations in order to plot error bars on the appropriate plots. Equation 3.18 is the general equation used for error propagation [121].

\[
\delta q = \sqrt{\left(\frac{\partial q}{\partial a} \delta a\right)^2 + \ldots + \left(\frac{\partial q}{\partial z} \delta z\right)^2} \tag{3.18}
\]

In this equation \(\delta_q\) is the error in a quantity \(q\) that is a function of \(a, b, \ldots, z\) param-
eters. The partial derivatives in the equation are the derivatives of the quantity with respect to each parameter it is a function of. The partial derivative is then multiplied by the error in that parameter and then added in quadrature with all other parameters in which \( q \) is a function of.

An example of how error is propagated through a velocity measurement will be demonstrated. The velocity is measured in the tunnel with static and total pressure transducers and a temperature probe using Bernoulli’s principal. This then leads to equation 3.19 which shows velocity as a function of the total and static pressure difference \((\Delta P = P_o - P_{stat})\) and fluid density \((\rho)\).

\[
v = \sqrt{\frac{2\Delta P}{\rho}} \tag{3.19}
\]

Using equation 3.18, the error in velocity is written as seen in equation 3.20 and the partial derivatives of this equation must be found and are displayed in equations 3.21, 3.22 and 3.23.

\[
\delta v = \sqrt{\left( \frac{\partial v}{\partial P_o} \delta P_o \right)^2 + \left( \frac{\partial v}{\partial P_{stat}} \delta P_{stat} \right)^2 + \left( \frac{\partial v}{\partial \rho} \delta \rho \right)^2} \tag{3.20}
\]

\[
\frac{\partial v}{\partial P_o} = \frac{1}{\sqrt{2} \rho \sqrt{\frac{P_o - P_{stat}}{\rho}}} \tag{3.21}
\]

\[
\frac{\partial v}{\partial P_{stat}} = -\frac{1}{\sqrt{2} \rho \sqrt{\frac{P_o - P_{stat}}{\rho}}} \tag{3.22}
\]

\[
\frac{\partial v}{\partial \rho} = -\frac{\sqrt{2} \Delta P}{2 \rho^{3/2}} \tag{3.23}
\]

These values were calculated for each angle of attack, multiplied by the error in pressure and density mentioned above and then added in quadrature per equation 3.20. This equation then provided the value for the velocity error and an error bar was added to the plot based on this value. The error analysis for all force data that were plotted
and in this work was done in this manner. Cavitation error was estimated by the range of observed data for each angle of attack. This was done due to the low number of realizations and the error in the pressure and velocity were not accounted for. This is a practice which was previously used before by ARL cavitation error analysis. Error bars were added to all force and cavitation plots and if they cannot be seen it is because they are within the size of the data marker.

3.5 Reynolds Number Effects

The full scale Reynolds number for this foil is 1.5 million and was able to be approximately achieved in this study. Some aspects of the testing resulted in higher Reynolds numbers than the design, specifically the cavitation testing. At lower angles of attack where cavitation was difficult to achieve, the tunnel velocity was increased when the tunnel pressure reached its lowest point in order to induce cavitation. The range of Reynolds numbers tested was $9.5 \times 10^5$ and $2.2 \times 10^6$. The effect of Reynolds number on the lift and drag is well known and documented for many airfoils as seen in “Theory of Wing Sections” by Abbott and Von Doenhoff [82]. A study performed by Jacobs and Sherman [104] was performed that tested the NACA 4412 over a wider range of Reynolds numbers than Abbott [82]. The NACA 4412 lift performance is illustrated in Figure 3.6, and for this particular foil generally as the Reynolds number is increased, the maximum lift coefficient increases as well. It can be seen that as the Reynolds number increased, the L/D also increased. This is because as the Reynolds number increases, the flow stays attached longer and therefore reduces the pressure drag on the foil.

The force data taken in the ARL 12 inch water tunnel was taken at the same Reynolds number and forces were not measured at higher Reynolds numbers because the dummy load cell was installed.

Reynolds number effects become more important when roughness is added to the foil because as Reynolds number increases, the boundary layer thickness decreases and the
Figure 3.6: Lift and L/D curves for the NACA 4412 over a large Reynolds number range. [104].

Roughness has more of an effect on the boundary layer [111]. Figures 3.7a and 3.7b show the Reynolds number effect on a foil with surface roughness.

Figure 3.7: Influence of Reynolds number on surface roughness effects. [111]
In order to examine the effects of the Reynolds number range run in these tests, the cavitation performance was tested at 1.6 and 2.3 million Reynolds number. Testing cavitation at Reynolds numbers lower than 1.6 million proved difficult due to the absence of cavitation. Figure 3.8 shows the variation in cavitation number due to Reynolds number difference experienced during these tests. Desinence cavitation was the cavitation calling criteria used for Figure 3.8. It can be seen that there is very little variation in cavitation number for this Reynolds number range.

Figure 3.8: Reynolds number effect on cavitation number on the MHKF1-180s foil with isolated roughness elements.

3.6 LDV Wake Profiles

The data acquired using the LDV system in order to calculate drag was in the form of velocity profiles. These velocity profiles needed to be integrated in order calculate the momentum deficit and drag. An example wake profile for the MHKF1-180s foil at negative four degrees angle of attack can be seen in Figure 3.9.

The integral used to calculate the drag coefficient can be seen in the equation 3.24. In this equation $c$, $v_0$, $v_{wake}$, and $z$ are the chord, free stream velocity, wake velocity and probe position respectively. The wake velocity is the velocity acquired by the LDV profile. The free stream velocity is taken as the average of the two wake end points. This
Figure 3.9: Wake profile for the MHKF1-180s at -4 degrees angle of attack.

is done in order to correct for blockage. This procedure was repeated for each angle of attack.

\[ c_d = \frac{1}{1/2cv_o^2} \int v_{\text{wake}}(v_o - v_{\text{wake}})dz \]  

(3.24)
Test Results

4.1 Test Plan

The tests performed in this work can be broken into three main groups which are force tests, cavitation calls and LDV profiles. Within these groups were tests conducted on the NACA 4412 foil and test conducted on the MHKF1-180s and MHKF1-180c foil. The MHKF1-180s and NACA 4412 were tested in a clean and rough configuration. Aside from these main tests were diagnostic tests to visualize flow patterns and assure proper tunnel operation. Table 4.1 is a table of the tests performed in this study.

4.2 Uniform Inflow Study

It was of interest to perform an LDV inflow study to verify the uniformity of the inflow velocity. The LDV system was used to traverse the test section in both width and height. The width direction is the horizontal direction across the test section perpendicular to the foil pressure and suction surfaces (20 inches) and the height direction is the vertical direction along the span of the fin (4.5 inches). The horizontal inflow profile was taken along the center line of both axes of the tunnel with no fin installed. The vertical surveys were taken 2.5 inches in front of the MHKF1-180s fin at 4 locations which were 3, 6,
Table 4.1: Test Matrix

<table>
<thead>
<tr>
<th>Test</th>
<th>NACA 4412</th>
<th>MHKF1-180s</th>
<th>MHKF1-180c</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clean configuration</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Forces</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Cavitation Call</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Cavitation breakdown</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LDV wake survey</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>LDV trailing edge survey</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Paint flow visualization</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Yarn tuft visualization</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>60 grit distributed roughness configuration</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Forces</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Cavitation Call</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Cavitation breakdown</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>46 grit isolated roughness configuration</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cavitation Call</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>16 grit isolated roughness configuration</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cavitation Call</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Trip wire configuration</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Forces</td>
<td>x</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

10 and 13 inches with respect to the inside of the tunnel window. The location of 2.5 inches was chosen because that was the furthest forward location optical access could be maintained for all four positions. A diagram of the vertical traverse locations can be seen in Figure 4.1 showing a top down view of the hydrofoil in the test section. It can be seen from Figures 4.2 and 4.3 that the inflow velocity in both directions is uniform within the range the tunnel impeller controller can hold. The vertical traverse shows lower velocities as you approach the fin due to blockage effects.
4.3 NACA 4412 Test Validation

4.3.1 Force Results

The experimental lift and drag results obtained from the NACA 4412 baseline study are shown in Figures 4.4 and 4.5 respectively. All data were corrected for tunnel block-
age based on standard wind tunnel wall interference methods presented in Barlow et al. [109]. Corrections were also made due to the flow introduced by the gaps in between the three sections which may have contaminated the two-dimensionality of the flow and must be subtracted out. The gap corrections were based on a hydrofoil end gap experiment by Kermeen [94] in which a NACA 4412 hydrofoil was tested and the effect on the forces was evaluated as a function of gap size. This data was used and extrapolated to correct for the gaps present in this study.

Figure 4.4: NACA 4412 lift data comparison with historic data.

Figures 4.4 and 4.5 show that the corrected experimental data agrees well with the
Figure 4.5: NACA 4412 drag data comparison with historic data.

historic water tunnel data [94] as well as the wind tunnel data taken in the NACA two-dimensional low-turbulence wind tunnel at 1 million Reynolds number [106]. The data taken from the NACA variable density tunnel [105] shows a variation in lift curve slope as well as $C_{l_{\text{max}}}$. The effect of turbulence intensity, tunnel conditions, the way the data was taken or a strong sensitivity to Reynolds number at these relatively low Reynolds number values may cause this variation in data. Both the ARL and Cal-Tech water tunnels have a turbulence intensity of approximately 0.2 to 0.3 percent, while the NACA low turbulence tunnel was an order of magnitude lower at levels of approximately a few hundredths of a percent [123, 110] and the NACA variable density wind tunnel was approximately 2 percent [111] which is relatively high compared to the other tunnels. It is shown in a work by Stack [103] that increasing turbulence intensity generally decreases the lift curve slope however, overall the work shows that the effect of turbulence intensity is foil dependent. A work by Fuglsang and Bove [108] draws the conclusion that two-dimensional tunnel testing is no trivial matter and involves more than just applying wall corrections. Details such as how the data was taken and model aspect ratio affect the results and it is stated that comparing results from different tunnels involves higher
uncertainties than wall corrections [108]. This work also showed that once stall and separation are present in the flow, two-dimensionality no longer exists [108]. The solid and three-part NACA 4412 foils also agree well proving the three-part fin concept is adequate for producing good force data.

The same good agreement can be seen in the drag data where historical data exists. In order to better compare the drag it may be easier to compare with the L/D plot which can be seen in Figure 4.6. It is seen that good agreement exists at $c_l$ values from -0.5 to 1.1, which is where the Cal-Tech and NACA low turbulence tunnel data ends. The NACA variable density tunnel however extends to higher $c_l$ values and a discrepancy can be seen which may be due to the same reasons stated above.

![Figure 4.6: NACA 4412 l/d data comparison with historic data.](image)

Based on the good agreement between the Cal-Tech water tunnel data and the NACA low turbulence tunnel it was concluded that the test set-up and methods were adequate to capture force data.
4.3.2 Cavitation Results

The cavitation performance was also verified against two separate historical data sets, both taken in the Cal-Tech water tunnel [94, 95]. The corrected experimental data can be seen compared to the historical data in Figure 4.7.

![Figure 4.7: NACA 4412 desinent cavitation performance compared with historical water tunnel data. Error bars represent the range of observed data calls.](image)

It can be seen from Figure 4.7 that the experimental data aligns well with both historic data sets, especially at the positive angles of attack. The type of cavitation that was seen during the cavitation call was traveling bubble cavitation and leading edge sheet cavitation once higher angles of attacks were reached. Cavitation in the gaps was also present, but this cavitation was ignored during the cavitation call and often disappeared before the traveling bubble or sheet cavitation. Air content differences, surface impurities and other parameters can effect nucleation and bubble formation resulting in data variability. The air content of the tunnel for the tests illustrated in Figure 4.7 was 4 ppm and the historical data did not specify the air content when the test was run. Desinent cavitation was called on the foil during this test as well as in both historical data sets, meaning when all the cavitation disappeared from the blade was when cavitation was called and the point was plotted. This was done three times for each angle of attack in order to obtain an average. The data agreed well with historical data therefore
validating the methods and hardware used to call cavitation.

The phenomenon of hysteresis was also analyzed on this foil in the distributed and isolated roughness conditions. The clean condition did not show any hysteresis effect. Figure 4.8 shows that when 60 grit distributed roughness is added to the leading edge of the foil there is minimal hysteresis effect. The isolated roughness conditions seen in Figures 4.9 and 4.10 show significant hysteresis effect. It is interesting that the 46 grit roughness size shows a much larger effect than the 16 grit. Both tests were run at the same air content of 3.4 ppm. When calling cavitation it was noticed that the incipient cavitation was not very consistent and varied substantially from run to run. For example certain runs would drop far below the desinence cavitation number than explode into fully developed cavitation.

Figure 4.8: NACA 4412 cavitation hysteresis with 60 grit roughness applied to the leading edge.
Figure 4.9: NACA 4412 cavitation hysteresis with 46 grit isolated roughness applied to the foil.

Figure 4.10: NACA 4412 cavitation hysteresis with 16 grit isolated roughness applied to the foil.

Cavitation Breakdown

Cavitation breakdown in both the clean and distributed roughness conditions was studied on the NACA 4412 three part foil. The air content for this test was 10.9 ppm. The roughness condition was 60 grit distributed roughness covering the first 7% of the chord. The cavitation numbers were corrected for blockage using methods presented in barlow et al.[109]. The NACA 4412 clean lift and drag breakdown data are presented graphically in Figures 4.11 and 4.12 while the data with 60 grit distributed roughness is illustrated by Figures 4.13 and 4.14. The data is presented as the ratio of the cavitating force coefficient to noncavitating condition.
Shen and Dimotakis [102] performed a similar study on a NACA 66 (mod) hydrofoil with an aspect ratio of 1.0 at a Reynolds number of 2 million. An increase in both lift and drag was noticed with an increasing cavity length (decreasing $\sigma$) up to 83% of the chord. According to this paper, two types of cavitation effected the lift results in two different ways. The two types of cavitation were leading edge sheet cavitation and mid-chord cavitation. Both forms of cavitation increased the lift by causing a lower than normal pressure due to the vaporization of liquid on the suction side. The leading edge sheet cavitation had a larger effect on the lift because there was a larger area in which the pressure was lower than normal. The mid-chord cavitation also increased the lift, but the effect was attenuated due to reduction in the leading edge pressure spike caused by a thickness effect from the cavitation. The drag increases for both types of cavitation due to a substantial increase in form drag caused by the cavitation [102].

The increase in lift due to cavitation can be seen in the clean condition for all angles of attack except for the 11 degree case. For most of the clean condition cases the increase in lift is quite small but the over all degradation of lift after the cavitation number decreases further is large. The effect on drag in the clean condition is more profound as seen in Figure 4.12. An interesting result is that when the cavitation number decreases to the point where the cavity extends past the trailing edge of the foil (super-cavitation) a reduction in drag can be seen.

In the 60 grit distributed roughness condition the increase in lift cause by cavitation can be seen more clearly. This is because in a non-cavitating flow the roughness reduces the lift so when cavitation occurs it effectively reduces the roughness and compared to the clean condition the increase in lift is larger. The leading edge roughness also insure that all cavitation is leading edge sheet cavitation therefore maximizing the increase in lift. The drag data for the roughness condition does not resemble the clean data at all. It can be seen that the presence of cavitation reduces the drag as the cavitation number decreases and the cavity grows. This is because the cavitation is making the roughness less effective, therefore reducing the skin friction drag. A point is reached however where
the form drag increase, mentioned above, outweighs the reduction in skin friction drag causing an overall drag increase seen as the inflection points on Figure 4.14.
4.3.3 Flow Visualization

Paint flow visualization was performed on the NACA 4412 to verify the flow was two-dimensional while still attached. It is known from Fuglsang and Bove [108] that when the flow separates, the two-dimensionality assumption is no longer valid because separation is a three-dimensional phenomenon. Illustrated in Figures 4.15 and 4.16 are the NACA 4412 at 0 and 15 degrees angle of attack at a Reynolds number of 1.3 million. Unfortunately
for the 5 degree visualization, two tone paint was not used yet, however the results can still be seen that the flow remains attached and is two-dimensional. The 15 degree case shows separation just past the mid chord point and the three dimensional effects of reversed and radial flow can be seen.

Figure 4.15: Oil paint flow visualization at 5 degrees angle of attack and 1.3 million Reynolds number showing two-dimensional flow.

Figure 4.16: Oil paint flow visualization at 15 degrees angle of attack and 1.3 million Reynolds number showing separated flow and loss of two dimensionality.
4.4 MHKF1-180 Hydrofoil Design Test Results

The tests conducted on the MHKF1-180s fin were done using the same proven methods as the NACA 4412 foil. The force results were also compared with the results from CFD simulations performed by The University of California-Davis. After the testing was complete on the MHKF1-180s, the trailing edge was modified from a curve to a flat bevel in order to study another anti-singing geometry. Force tests and more importantly trailing edge LDV studies were performed on this modified foil to compare with the curved anti-singing geometry.

4.4.1 Force Results

All force data were corrected for tunnel interference effects using the same methods found in Barlow et al. [109] as previously mentioned. The experimental lift and drag results in the clean condition shown in Figures 4.17 and 4.18 compare the MHKF foils with NACA 4412 and NACA 4418 foils. It was observed that the MHKF foils had higher overall lift throughout the entire angle of attack range. When comparing the NACA 4412 experimental data to the thicker NACA 4418 Abbott data [82], it was seen that slope is the same in the linear region of the lift curve but the NACA 4418 has a higher maximum lift. It is worth mentioning that in the Abbott data collected on the NACA 4418, there was little Reynolds number dependence in the 3 million to 9 million range tested while the NACA 4412 showed a larger Reynolds number dependence [82]. This Reynolds number dependence is a function of airfoil thickness with thicker foils being less Reynolds number dependent [82]. When Comparing the MHKF1-180s to the MHKF1-180c, it can be seen that the MHKF1-180c had slightly higher lift before stall than the MHKF1-180s.

When analyzing the clean condition experimental drag data it was seen that the MHKF1-180s had comparable drag with the NACA 4412 within the error of the load cell except for large negative angles of attack. The Abbott NACA 4418 data is much lower at
angles of attack where separation occurs. The MHKF1-180c had significantly lower drag than the MHKF1-180s and NACA 4412 foils, which translated into the MHKF1-180c being a more hydrodynamically efficient design based on the lift to drag ratio as seen in Figure 4.19. This satisfies one of the design criteria of high lift to drag ratio. The MHKF1-180s also shows increased L/D performance when compared to the NACA 4412 above $c_l$ values of .8. The increase in performance of the MHKF1-180c compared to the MHKF1-180s is larger than the increase in performance of the MHKF1-180s compared to the NACA 4412.

Figure 4.17: MHKF1-180s, MHKF1-180c, NACA 4412 and NACA 4418 lift compared in the clean condition.
Figure 4.18: MHKF1-180s, MHKF1-180c, NACA 4412 and NACA 4418 drag compared in the clean condition.

Figure 4.19: MHKF1-180s, MHKF1-180c, NACA 4412 and NACA 4418 L/D comparison in the clean condition.
The experimental results showing the roughness sensitivity for the MHKF1-180s and the NACA 4412 are illustrated in Figures 4.20, 4.21 and 4.22. In these figures clean, distributed roughness and trip wire conditions as well as NACA 4412 clean and rough data are displayed. The drag data are also compared to the LDV wake profile data. The distributed roughness used was 60 grit carborundum applied to the first 7 percent of the leading edge. The trip wire was installed at the 7 percent chord location.

Figure 4.20: MHKF1-180s and NACA 4412 lift compared at several roughness cases at 1.3 million Reynolds number.

When analyzing the addition of distributed roughness and the sensitivity of the forces it can be seen that the roughness reduces the maximum lift and lift curve slope while increasing the drag for both foils, which is an expected result due to the premature transition of the boundary layer effectively de-cambering the foil [101, 83]. The trip wire on the MHKF1-180s has the same effect on lift as the distributed roughness, but is less severe. This is due to the distributed roughness tripping the boundary layer immediately at the leading edge whereas the trip wire allows the boundary layer to develop until the wire is reached therefore the de-cambering effect is less severe. The trip wire increases
Figure 4.21: MHKF1-180s and NACA 4412 drag data compared at several roughness cases at 1.3 million Reynolds number.

Figure 4.22: MHKF1-180s L/D data at 1.3 million Reynolds number.
the drag from -5 to 7 degrees but decreases the drag at angles both lower and higher than this range. This is believed to be due to the turbulent boundary layer allowing flow to stay attached longer and therefore reduce the pressure drag. The LDV computed drag data also agrees with the load cell data.

The roughness sensitivity comparison between the MHKF1-180s and NACA 4412 showed that the NACA 4412 has less of a sensitivity to roughness as seen in Figure 4.23. The roughness sensitivity is represented as the delta L/D which is the difference between the L/D in the clean condition and the L/D in the rough condition for each angle of attack. This increased sensitivity to roughness could again be contributed to the foil not behaving as designed with an extended region of laminar flow. The premature tripping of the boundary layer would have a large effect on the performance which we may be observing. Also the NACA 4412 is a 12 percent thick foil while the MHKF1-180s is 18 percent thick. It is documented in literature [82] that thicker foils have more of a sensitivity to roughness which contributes increased roughness sensitivity of the MHKF1-180s foil.

In order to justify this, The Abbott [82] data for the NACA 4418 and NACA 4412 was compared. This was done because the NACA 4418 has the same thickness ratio which better represents the MHKF1-180s foil. Figure 4.24 shows the L/D plot comparing these two foils. It can be seen from this plot that in the clean condition the NACA 4412 has slightly better L/D performance compared to the NACA 4418. When the roughness was added it can be seen from the figure that the change in L/D is much greater for the NACA 4418 when compared to the NACA 4412. The MHKF1-180s foil was not added to this figure because the roughness applied to the MHKF1 and NACA 4412 during the experiment was more severe than the NACA standard roughness in order to better simulate real operating conditions. The NACA standard roughness was a roughness height to chord ratio of $4.58 \times 10^{-4}$ spread over the first 8 percent of the leading edge covering an area of 5 to 10 percent. Comparatively the roughness applied to the MHKF1-180s and NACA 4412 during the experiments had a roughness height to chord ratio of
$1.25 \times 10^{-3}$ spread over the first 7 percent of the leading edge covering roughly 50 percent of the area. With these results from Abbott it can be seen that thickness effects the roughness sensitivity and therefore comparing the MHKF1-180s with the NACA 4412 may not be an adequate evaluation of foil performance. Somers [124] also performed a study on several natural laminar flow wind turbine airfoils and concluded that as thickness increased the sensitivity to roughness also increased.

Figure 4.23: MHKF1-180s and NACA 4412 roughness sensitivity comparison.
The results illustrate the adverse effect of roughness on foil performance if debris or biofouling accumulates on the blade during operation underwater. The accumulation of debris on the foil could also cause premature surface cavitation which is addressed in the next section.

### 4.4.2 Cavitation Results

Desinent cavitation calls in the clean condition were performed on both the NACA 4412 and MHKF1-180s foils and compared in Figure 4.25. The MHKF1-180s cavitation assessment was done at an air content of 3.1 ppm and the NACA 4412 assessment was done at a similar value of 4.0 ppm. The MHKF1-180s foil shows improved cavitation performance over the NACA 4412 at higher angles of attack. Figure 4.26 shows developed cavitation comparison between the NACA 4412 and MHKF1-180s foils at the same cavitation number and angle of attack. Notice the three different types of cavitation on the solid NACA foil in Figure 4.26a. Traveling bubble, sheet and gap cavitation are all labeled in the figure. When comparing the MHKF1 foil it can be seen that there is a
significant amount of gap cavitation present due to the gaps separating the three part fin. Ignoring this however it can be seen that the cavitation present on the fin is less severe than the cavitation on the NACA 4412.

![Figure 4.25](image1.png)

Figure 4.25: Foil clean desinent cavitation performance comparison.

![Figure 4.26](image2.png)

Figure 4.26: Photographs comparing developed cavitation on the NACA 4412 and MHKF1-180s at the same condition.

The phenomenon of hysteresis was also analyzed on the MHKF1-180s foil in the distributed and isolated roughness conditions. The clean condition had very little hysteresis effect as seen in Figure 4.27. Figure 4.28 shows the effect 60 grit distributed roughness has on cavitation hysteresis and it can be seen that while the values do not differ greatly, the incipient cavitation number is higher than the desinent cavitation number. This is
interesting because the desinent cavitation number is generally the more conservative case therefore it is historically higher than the incipient cavitation number. The 46 and 16 grit isolated roughness hysteresis plots can be seen in Figures 4.29 and 4.30. It can be seen that there is no hysteresis effect for the 46 grit isolated roughness case and the 16 grit roughness case does not show a large variation at lower angles of attack, but once higher angles are reached and separation is a concern the difference becomes larger. In this case the incipient cavitation number is lower than the desinent which agrees with typical trends in historical data.

Figure 4.27: MHKF1-180s cavitation hysteresis under the clean condition.

![Figure 4.27](image1)

Figure 4.28: MHKF1-180s cavitation hysteresis with 60 grit roughness applied to the leading edge.

![Figure 4.28](image2)

The cavitation sensitivity to roughness of the NACA 4412 and the MHKF1-180s studied and the results can be seen in Figures 4.31 and 4.32 respectively. The roughness
added was 60 grit distributed roughness to the first 7 percent of the leading edge, 46 grit isolated roughness and 16 grit isolated roughness at select chord locations. The distributed roughness cavitation call was done at an air content of 4.3 ppm and the isolated roughness call was done at 3.3 ppm for both foils. It is interesting to see that the addition of distributed roughness to the leading edge of the NACA 4412 foil only marginally effects the cavitation performance while the cavitation performance of the MHKF1-180s was decreased more noticeably. The sensitivity is portrayed more clearly in Figure 4.33. This plot shows the delta $\sigma$ vs. angle of attack. The delta $\sigma$ is the difference between the roughness cavitation and the clean cavitation for each roughness
case. The blue and red diamonds show that while there is not much difference in the sensitivity there is some, especially in the intermediate angle of attack range where this foil is likely to operate. This shows that the NACA 4412 is less sensitive to distributed roughness cavitation than the MHKF1-180s foil.

The addition of isolated roughness had a much larger effect on the cavitation performance of both foils which is an expected result based on a work by Arndt et al. [100]. The results for the 46 grit isolated roughness elements is represented by the squares in Figure 4.33 and it can be seen that the NACA 4412 sensitivity is initially lower, but once an angle of approximately 7 degrees is reached the foil becomes much more sensitive than the MHKF1-180s. When the roughness size is increased to 16 grit represented by the triangles on Figure 4.33 There is initially little difference in the sensitivity between the two foils, but once the angel of approximately 5 degrees is reached the MHKF1-180s becomes more sensitive up until an angle of approximately 10 degrees is reached where the NACA 4412 becomes more sensitive. It is worth noting that the delta $\sigma$ for the NACA 4412 for both the 46 and 16 grit size elements is similar at angles above approximately 10 degrees. This shows that the size of the roughness does not have a sizeable impact in regions near stall for the NACA 4412. The NACA 4412 insensitivity to roughness may be due to the larger pressure peak due to the smaller leading edge radius. While the MHKF1-180s showed better performance in a clean condition when compared to the NACA 4412, the NACA 4412 showed better sensitivity to roughness in most scenarios. It is difficult to make any concrete conclusions from this data set because the roughness elements were not all the same size or shape and the critical cavitation number is dependent on the minimum pressure at the top of the roughness element [101]. This means that in the MHKF1-180s test the isolated roughness element at a given location could have been rounded while in the NACA 4412 test at the same location it could have been tall and spiky. This is less of a concern with the distributed roughness which is why the sensitivities are more closely related. The same thickness effect present in the force sensitivity to roughness may also have an effect in the cavitation performance. The
circles labeled by percent chord locations refer to the specific roughness element that was the last to cavitate in Figures 4.31 and 4.32.

Figure 4.34 shows cavitation occurring on both distributed and isolated roughness on the MHKF1-180s at a similar condition of 8 degrees and an approximate $\sigma = 1.0 - 1.2$. 
Figure 4.31: NACA 4412 desinent cavitation performance with roughness elements added.

Figure 4.32: Isolated and distributed roughness effect on desinent cavitation number on the MHKF1-180s foil.
Figure 4.33: Cavitation roughness sensitivity comparison for the MHKF1-180s and NACA 4412.

(a) MHKF1-180s distributed roughness cavitation at $\sigma = 1.1$ and $\alpha = 8$ degrees.

(b) MHKF1-180s isolated roughness cavitation at $\sigma = 1.2$ and $\alpha = 8$ degrees.

Figure 4.34: Comparison of distributed and isolated roughness on the MHKF1-180s at a similar condition.

Cavitation Breakdown

The cavitation breakdown test was performed at an air content of 8.4 ppm in both the clean and distributed roughness condition. The data is presented as the ratio of the cavitating lift and drag coefficient to the non-cavitating lift and drag coefficient as seen in Figures 4.36 and 4.37. Shen and Dimotakis performed a study on the influence of surface cavitation on hydrodynamic forces [102] and the trends in their data match
the experiment data. The increase in lift can be seen for all angles except 7 degrees. Once the cavity covered a significant portion of the fin the lift decreased drastically with continually decreasing cavitation number. The drag behavior is different in that as the cavitation number decreases, the drag initially increases due to added form drag but then decreases with further decreasing cavitation number as the foil approaches the super-cavitating regime. This drag reduction is due to the foil being enveloped in a pocket of air instead of water reducing skin friction. An example of what the cavitation may look like when forces breakdown can be seen in Figure 4.35. The red dotted lines in this figure represent the range the cavity length fluctuated and is used to emphasize the unsteadiness of the cavity length. This specific example is the MHKF1-180s foil with 60 grid distributed roughness applied.

![Figure 4.35: MHKF1-180s example of cavity length when breakdown is about to occur. The red lines represent the unsteadiness of the cavity length.](image)

The roughness force breakdown plots can be seen in Figures 4.38 and 4.39. The lift breakdown with added distributed roughness followed the same trends as the NACA 4412 foil where the lift increased and then once the cavity covered a large portion (appx. 80 percent) of the foil, the lift drastically decreased. The drag on the other hand did not follow the same trend whatsoever. The NACA 4412 foil showed a decrease in drag
due to cavitation rendering the roughness less effective then increase when the form drag became sufficiently large. The MHKF1-180s foil shows similar behavior to the clean condition which is a very interesting result.

Figure 4.36: MHKF1-180s cavitation lift breakdown

Figure 4.37: MHKF1-180s cavitation drag breakdown
4.4.3 Flow Visualization

In order to visualize and verify two-dimensional flow, oil paint was used to characterize the flow patterns present on the foil. Visualizations were done at 5° and 15° degrees to show both attached and separated flow. As stated above from the work of Fuglsang, two-dimensionality no longer exists in separated flow [108] which is verified in Figure 4.40a. Flow was visualized during an attached flow configuration at 5 degrees as well which
shows two-dimensional flow is maintained seen in Figure 4.40b.

(a) MHKF1-180s paint flow visualization at 15 degrees showing 3 dimensional flow caused by separation.

(b) MHKF1-180s paint flow visualization at 5 degrees showing 2 dimensional flow is maintained.

Figure 4.40: Paint flow visualization for separated and attached flow at 1.3 million Reynolds number.
The MHKF1-180s foil that was tested in the water tunnel was manufactured with a curved anti-singing trailing edge. Because of this, the MHKF1-180 with a sharp trailing edge geometry could not be tested in the water tunnel and the effect the anti-singing edge had on performance could not be evaluated experimentally, however Shiu et al. [72] performed CFD simulations the MHKF1-180 without any anti-singing geometry, the MHKF1-180s with a curved anti-singing geometry and the MHKF1-180b with a beveled anti-singing geometry. It was found that the performance impacts of the anti-singing geometries had small impacts on the hydrodynamic performance. The MHKF1-180b is similar to the MHKF1-180c with the difference being the MHKF1-180b has a bevel that starts at same point the curve starts for the MHKF1-180s while the MHKF1-180c is a modified version of the MHKF1-180s with a cut to match the angle of the bevel of the MHKF1-180b.

The LDV spectra data for both variations of the MHKF1-180 foil and the NACA 4412 are presented in this section. The velocity data acquired by the LDV system were post processed to calculate the spectra, and then non-dimensionalized. The length scale, free stream velocity, maximum RMS velocity and Strouhal number were the quantities used to non-dimensionalize the data and the non-dimensional quantities can be seen in Figures 4.43 to 4.46. The maximum RMS velocity as well as the wake depth and width are presented in Table 4.2. The length scale of this analysis was the width of the trailing edge and this was chosen due to previous work presented in Blake [73].

<table>
<thead>
<tr>
<th>Foil</th>
<th>Thickness (mm)</th>
<th>vRMS (m/s)</th>
<th>Wake depth (m)</th>
<th>Wake width (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NACA 4412</td>
<td>0.5</td>
<td>0.7</td>
<td>2.42</td>
<td>0.009</td>
</tr>
<tr>
<td>MHKF1-180s</td>
<td>5.08</td>
<td>2.24</td>
<td>5.88</td>
<td>0.009</td>
</tr>
<tr>
<td>MHKF1-180c</td>
<td>5.08</td>
<td>1.62</td>
<td>5.23</td>
<td>0.009</td>
</tr>
</tbody>
</table>

The spectral data is presented in Figures 4.43 to 4.46 for the four different velocities.
tested. It can be seen from the spectra that there is a noticeable spike in the data for the MHKF1 foils, especially the MHKF1-180c. It is believed that this spike is associated with the "knuckle" of the anti-singing trailing edge and can be related to the Strouhal number using a length scale of the anti-singing trailing edge thickness. The frequency was found using the equation for the Strouhal number seen in equation 4.1 where $S_t$ and $d$ are the Strouhal number and length scale respectively.

$$f = \frac{S_t v_\infty}{d} \tag{4.1}$$

The Strouhal number used in this equation was approximated at .21 based on the Strouhal number of an infinite circular cylinder at a Reynolds number of 1 million [125]. The comparison of the frequencies at the peaks in the experiment and the frequencies calculated using this method can be seen in Figure 4.42.

It can be seen that the calculated values are comparable to the experimental values which gives reason to believe the frequency spikes are due to the anti-singing trailing edge. It is likely that a frequency spike is not seen in the NACA 4412 foil data because the survey was taken too far down stream of the trailing edge. The non-dimensional distance downstream ($x_{TE}/d$) was 10 for the NACA 4412 and 1 for the MHKF1-180 foils where $x_{TE}$ and $d$ are the distance downstream from the trailing edge and the trailing edge.
thickness respectively. This puts the LDV probe in the far wake for the NACA 4412 while the MHKF1-180 it is in the near wake thus why no spike is seen in the NACA 4412 data.

This LDV shedding data however is not enough to quantify the acoustic signatures of the foils. Trailing edge singing is a fluid structure interaction and the response of the structure to the trailing edge vortex shedding needs to be determined in order to draw any conclusions on improved acoustic signature due to trailing edge geometry. This may be done by measuring the trailing edge vibration using a laser vibrometer. This is considered future work for this project.
Figure 4.43: Trailing edge LDV spectra data comparing the three foils at 5 meters per second.

Figure 4.44: Trailing edge LDV spectra data comparing the three foils at 6 meters per second.
4.6 CFD Comparison

The University of California - Davis performed the CFD analysis for this project using the CFD tool OVERFLOW. An independent CFD analysis was also performed in Fluent and OpenFOAM parallel to this by The Pennsylvania State University for educational and data verification purposes. The grids at Penn State were generated using Pointwise meshing software and the simulations were run using Fluent version 6 and OpenFOAM.
Grid Generation

The CFD simulation performed was a two dimensional simulation and was run on both the NACA 4412 and MHKF1-180s foils. Two types of grids were generated in this study with the purpose of studying blockage effects caused by the tunnel walls. This was performed by first creating an O-grid surrounding the foil simulating an infinite domain and a rectangular grid that simulated the tunnel boundaries. The tunnel domain was comprised of a smaller more refined o-grid surrounded by tetrahedral mesh filling the rest of the space. An example of the O-grid domain and the tunnel domain can be seen in Figures 4.47 and 4.48. The angle of attack was changed in the tunnel domain by rotating the nested o-grid within the tetrahedral elements. The initial step size for the O-grid surrounding the foil was $1 \times 10^{-5}$ inches and the growth rate was 1.1. This initial step was chosen to be this size to capture the boundary layer and turbulence. All grid lines are not shown in the infinite case due to resolution.

![Figure 4.47: MHKF1-180s tunnel grid at 0 degrees angle of attack.](image)

Fluent Solver and Boundary Conditions

The O-grid was split in half down the middle and the front half, closest to the leading edge, was given an inflow boundary condition while the half on the trailing edge side was given an outflow condition. The airfoil surfaces were given a no slip wall boundary
The boundary conditions for the tunnel domain were inflow on the upstream vertical, outflow on the downstream vertical and no slip walls on the two horizontal tunnel walls and the airfoil surfaces.

Fluent was run in the two dimensional double precision mode and utilized a steady pressure based solver. The velocity inlet boundary condition had an x component velocity initialized at 6.069 m/s and a turbulence intensity and length scale set to 0.3% and .0001 meters respectively. The turbulence model used was a 2 equation k-w SST model with a built in transition model. The solution controls were set to couple pressure and velocity and all discretizations were set to second order. Convergence occurred when all residuals reached a value below $1 \times 10^{-5}$.
OVERFLOW Solver and Boundary Conditions

Both the NACA 4412 and MHKF1-180s foils were simulated in OVERFLOW Shiu [72]. Shiu et al. [72] provides more detail about the OVERFLOW analysis. OVERFLOW is a two or three dimensional, unsteady Reynolds averaged Navier Stokes solver with various turbulence models [126]. The turbulence model used was the k-w SST model and the grid used was an O-grid which simulated an infinite domain. Free transition was simulated by activating the turbulence model at the transition point given by XFOIL. The flow leading up to this point was laminar.
Experimental Results Comparison

Figures 4.49 and 4.50 show the comparison of the experimental water tunnel data taken in the ARL/PSU 12-inch diameter water tunnel, OVERFLOW results, Fluent results and the experimental data from the Cal. Tech. water tunnel [94]. It can be seen that both the lift curves produced by the CFD solvers have higher lift curve slopes and $c_{l_{max}}$ with the OVERFLOW results having the highest values. All results show the same zero lift angle of 4 degrees. The drag results show good agreement at angles where there is no separation, but once the angle of attack is increased past 10 degrees the results begin to diverge. This may be attributed to the computation inability to accurately model separation and stall.

![Figure 4.49: NACA 4412 lift comparison of CFD and experimental data.](image)

It is also seen that the Fluent and OVERFLOW results do not match for the NACA 4412 lift and drag. The sources of this discrepancy may be in the way transition was modeled or sensitivity to grid. A grid sensitivity study was run by UC - Davis and it was seen that the results do vary with changing grid.

The differences in the CFD and experimental data for the MHKF1-180s foil is larger than the NACA 4412 discrepancies as seen in Figures 4.51 and 4.52. It can be seen that the lift curve slope and $c_{l_{max}}$ for the Fluent, OVERFLOW and OpenFOAM CFD runs are all in good agreement but are higher than the experimental values. The OpenFOAM
run with the kkl-w transitional flow turbulence model [127] retains the same slope as all the other CFD results, but stalls significantly earlier and has a more comparable $c_{l_{max}}$ to experimental data. This particular model shows improved prediction of transition when compared to other turbulence models as reported in a work by Walters [127]. This discrepancy between turbulence models in the simulations may indicate that the foil is more sensitive to transition and stall than expected, or the turbulence model used may not be able to adequately capture these phenomena. The MHKF1-180s foil is designed to have an extended laminar flow [72] region and if that laminar flow is prematurely tripped to turbulent by free stream turbulence, tunnel or roughness effects it is likely that the performance will degrade due to unexpected de-cambering.

The drag data is in better agreement with the experimental results as seen in Figure 4.52. The Fluent and OVERFLOW results are in good agreement but the OpenFOAM results using the same k-w SST turbulence model as Fluent differed, especially at high angles of attack where separation and stall are likely. This result is interesting because the lift data agreement was very good between the three k-w SST simulations. The OpenFOAM result which utilized the kkl-w transitional model showed the best agreement with the experimental data just as it did in the lift case again indicating the
importance of properly capturing transition.

Figure 4.51: MHKF1-180s lift comparison of CFD and experimental data.

Figure 4.52: MHKF1-180s drag comparison of CFD and experimental data.

OVERFLOW Comparison with a NACA 4418

When the University of California Davis designed and performed their CFD analysis a NACA 4418 was used as a comparison because it matched the maximum thickness to chord ratio of the MHKF1-180s of 18 percent. According to Shiu et al. [72] when the MHKF1-180s CFD results were compared to the CFD results of the NACA 4418 the MHKF1-180s had a higher maximum $c_l$ value. The maximum lift to drag ratio
for the NACA 4418 was higher than the MHKF1-180s, but the MHKF1-180s retained much better soiled performance when compared to the NACA 4418 which is an important design criteria especially when operating in water. Figure 4.53 from Shiu et al. [72] shows the performance comparison predicted by the CFD simulations with XFOIL comparison as well.

Figure 4.53: OVERFLOW and XFOIL analysis of MHKF1-180s and NACA 4418 foil.[72]
Conclusions and Future Work

5.1 Summary and Conclusions

A hydrofoil was designed for application on a marine hydrokinetic turbine at the University of California - Davis. The design requirements for the foil were high lift to drag ratio to increase efficiency, low sensitivity to roughness, well defined stall point, low sensitivity to blade surface cavitation, anti-singing trailing edge to mitigate noise and vibration and sufficient thickness to withstand structural loads. Once a design was decided, a model was made to be tested in the PSU/ARL 12-inch diameter water tunnel. The test was a two-dimensional test. The model that was created was unique in the sense that it utilized a 3 piece design in which the end pieces were not connected to the load cell with the intent of reducing data contamination from end wall effects. In order to test this unique three piece fin design, both a solid and three piece NACA 4412 foil were created to verify the methods and hardware setup were adequate to produce good data. After comparing the data between the three piece and the solid foils it was concluded that the three piece design was adequate. The NACA 4412 foil was chosen as a baseline and for general performance comparisons because of the large historical database.

When testing began, diagnostic tests were run in order to verify the proper operation of the tunnel. A uniform inflow study of the rectangular test section was performed using
an LDV laser to measure velocity. The conclusions drawn from this study were that the velocities in both directions were uniform. A paint flow study was also performed on each foil to verify that the flow was two-dimensional in nature. The conclusions drawn for both foils was that the flow was two-dimensional while attached but the two-dimensionality was lost once separation occurred. The test setup and methodology was further verified by comparing the force and cavitation data of the NACA 4412 with several sets of historical data and the conclusion was that the experimental data taken in the 12 inch water tunnel was in good agreement with historical data.

With the setup and methodology verified the tests began on the MHKF1-180s foil. The force results showed higher $c_{\text{max}}$ and a well defined yet gentle stall for the MHKF1-180s when compared to the NACA 4412. This translated into better L/D performance when compared to the NACA 4412 satisfying 3 of the design criteria. When compared to the NACA 4418 data from Abbott [82] the MHKF1-180s showed an overall increase in performance in the linear region of the lift curve and had a comparable $c_{\text{max}}$. The MHKF1-180s also had a larger lift curve slope than both the NACA 4412 and NACA 4418 further improving performance. Force data was also acquired on the MHKF1-180c with the beveled trailing edge and it was concluded that this trailing edge geometry both reduced the drag and increased the lift therefore increasing the L/D performance.

The sensitivity to roughness was then analyzed and compared to the NACA 4412. When 60 grit carborundum elements were added to the foil leading edge, the lift curve slope and maximum lift decreased and the drag increased for both the NACA 4412 and MHKF1-180s foils. A trip wire was also installed on the MHKF1-180s foil and showed the same result of decreased lift slope and maximum lift, but showed a decrease in drag at higher angles of attack. It is thought this is due to the tripping of the boundary layer to turbulent reducing drag without the added skin friction of the distributed roughness elements. When the sensitivity to roughness was analyzed the conclusion drawn was that the MHKF1-180s forces were more sensitive to distributed roughness when compared to the NACA 4412. This increased sensitivity to roughness was likely due to increased
thickness causing increased sensitivity to roughness found in works by Abbott [82] and Somers [124]. This increase in sensitivity was an undesired result but comparing two foils of different thickness when it is known that a thickness effect is present is not a good gauge of performance.

The cavitation performance was analyzed and compared to the NACA 4412 and the conclusion was that under a clean configuration, the MHKF1-180s had a broader cavitation bucket which resulted in improved cavitation performance satisfying one of the design criteria. Hysteresis was also studied on the MHKF1-180s foil and it was found that cavitation hysteresis did not exist under the clean configuration and was very small with roughness added. The NACA foil also showed no hysteresis under the clean configuration, but when isolated roughness was added to the foil there was a large hysteresis effect. Cavitation performance sensitivity to roughness was also performed on both foils. The results were that with distributed roughness applied there was little difference between the sensitivity of the two foils, but when isolated roughness was applied the NACA 4412 was overall less sensitive to roughness when compared with the MHKF1-180s. This difference may also be due to the thickness effect on roughness sensitivity found in the force data. Also the critical cavitation number depends on the minimum pressure coefficient at the top of the roughness element, and the roughness elements may have not been uniform between tests. Future work is needed in this area and will be addressed in a later section.

The breakdown of the lift and drag forces due to cavitation was also analyzed and compared. For both the MHKF1-180s and NACA 4412 foils an increase in lift was the result of decreasing cavitation number up until approximately 80 percent of the chord was covered in cavitation. When the bubble extended further than this, the lift dramatically reduced with further reduction in cavitation number. Up until this point the drag remained relatively the same, but once this point was reached the drag increased dramatically with further decreasing cavitation number until a point was reached when the drag decreased again corresponding to the point when the cavitation bubble extended
past the trailing edge (super-cavitation). When roughness was added, the MHKF1-180s foil showed similar results as the clean condition, but the NACA 4412 showed a reduction in lift with decreasing cavitation number initially, then the drag began to increase but never to the non-cavitating level. It is believed that the cavitation forming over the roughness rendered the roughness less effective therefore reducing the overall drag until the form drag caused by the cavity bubble caused the drag to increase with further decreasing cavitation number. Why this result was not seen on the MHKF1-180s is subject for future work.

The spectral analysis of the anti-singing trailing edge vortex shedding was compared between the NACA 4412, MHKF1-180s and MHKF1-180c (bevel cut). The conclusion drawn was that there were spikes in both the MHKF1-180 foils at frequencies that corresponded with the Strouhal number of the anti-singing trailing edge thickness. The effect this has on noise is unknown at this point, but if the spike does not increase the noise of the foil, the MHKF1-180c is the best performing foil as stated prior. This experiment was added late to the project and to go into great depths of this analysis is beyond the scope of this thesis.

The University of California - Davis was responsible for the CFD analysis of the MHKF1-180s foil design, but a parallel analysis was performed at Penn State. The University of California - Davis ran their simulations using OVERFLOW with a k-w SST turbulence model. Transition was modeled by turning on the turbulence model at the chordwise location XFOIL calculated as the transition point. Fluent and OpenFOAM were used to run the simulations at Penn State and the turbulence model used in Fluent was a k-w SST with transition while OpenFOAM used a kkl-w transitional turbulence model that more accurately modeled transition [127]. The conclusions of this study were that both the NACA 4412 and MHKF1-180s CFD simulations had higher lift curve slopes and maximum lift coefficients as well as lower drag when compared to the experimental results. This translated into high L/D when compared to the experimental data. The kkl-w transitional OpenFOAM turbulence model was only run on the MHKF1-180s and
the comparison was the closest of any of the simulations. The lift curve slope was still significantly higher, but the maximum lift matched the experimental data better. This turbulence model still had lower drag, but better matched the experimental data at higher angles of attack. Comparing the kkl-w transitional turbulence model to the standard k-w SST gave reason to believe that either the CFD was not correctly capturing transition, or the foil was prematurely transitioning in the tunnel. The OpenFOAM standard k-w SST simulation agreed well with the experimental drag data which was an interesting result because the lift data for this transition model agreed with the other k-w SST simulations which was higher than the experimental lift data. OVERFLOW was also used to compare the MHKF1-180s to the NACA 4418 foil because this foil had the same thickness to chord ratio. The results of this study were that the MHKF1-180s had a higher maximum $c_l$ but lower maximum L/D when compared to the NACA 4418. The MHKF1-180s also had less sensitivity to roughness which was a desired design characteristic.

5.2 Future Work

The discrepancies in the comparison of the CFD and experimental results requires additional consideration. The reasons for this discrepancy are not well understood at this point, but it may be due to the MHKF1-180s foil being designed to have an extended region of laminar flow. When tested in the tunnel the region that was supposed to be laminar may have prematurely tripped to turbulent therefore changing the performance. It would be of interest to experimentally test where the flow transitioned on the MHKF1-180s foil and compare that location to XFOIL. It would also be of interest to make a solid 1 piece foil model for the MHKF1-180s to install in the tunnel and compare to the three part design. This would be beneficial because the gap flow present in the three part design may have tripped the boundary layer in the laminar region causing the reduction in lift. This result was not seen when comparing the NACA 4412 solid fin to the three
part model because the NACA 4412 was not designed in this manner. It is interesting however because the CFD results did not compare well with the experimental results found in this experiment or historically. It would be of interest to identify the root of this discrepancy.

It would also be of interest to test the roughness sensitivity again using uniform roughness elements, especially in the isolated roughness case. The fact that the roughness elements could have been a different size on one foil when compared to the other at the same chordwise location could be an explanation why the MHKF1-180s foil did not perform as well in the roughness sensitivity test. Glass beads of a uniform diameter could perhaps be used.

The results for the cavitation breakdown under the rough condition did not follow the same trends between the NACA 4412 and the MHKF1-180s and at this time it is not fully understood. It would be interesting to repeat this testing with the uniform roughness elements as well as different sized roughness elements.

The brief spectral analysis on the trailing edge performed in this study is not enough analysis to come to a conclusion on the acoustic performance of the MHKF1-180s. This subject could be a thesis in itself and the amount of analysis and testing that would be involved was beyond the scope of this work. It is of interest however to pursue future work in this area because acoustics is an important parameter in environmental limitation of marine turbines.
References


[113] Texas A&M University Aerospace Engineering Division, College Station, TX, Oran W. Nicks *Low Speed Wind Tunnel Facility Handbook*, September 2000.


