Computational Modeling of A Williams Cross Flow Turbine

Sajjan Pokhrel
Wright State University

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

Part of the Oil, Gas, and Energy Commons, and the Power and Energy Commons

Repository Citation
https://corescholar.libraries.wright.edu/etd_all/1876

This Thesis is brought to you for free and open access by the Theses and Dissertations at CORE Scholar. It has been accepted for inclusion in Browse all Theses and Dissertations by an authorized administrator of CORE Scholar. For more information, please contact library-corescholar@wright.edu.
COMPUTATIONAL MODELING OF A WILLIAMS CROSS FLOW TURBINE

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Renewable and Clean Energy Engineering

By

SAJJAN POKHREL
B.E., Tribhuvan University, 2013

Wright State University
2017

____________________________
James Menart, Ph.D.
The thesis Director

____________________________
Joseph C. Slater, Ph.D., P.E.
Chair
Department of Mechanical and Materials Engineering

Committee on Final Examination

____________________________
James Menart, Ph.D.

____________________________
Subramania I. Sritharan, Ph.D., P.E.

____________________________
George Huang, Ph.D.

____________________________
Barry Milligan, Ph.D.
Interim Dean of the Graduate School
ABSTRACT


Hydropower is not only the most used renewable energy source in the United States, but in the world. While it is well known that large hydropower facilities, like the Hoover Dam, provide large amounts of electrical power, there is also a tremendous opportunity for hydroelectric power generation from small scale facilities that has largely been overlooked. The work being presented here studies a new cross flow turbine called the Williams Cross Flow Turbine (WCFT), which was designed to extract electric energy from low head, run-of-the-river, small hydropower sites.

To spur the implementation of the WCFT in small hydropower applications, and thus to spur the development of small hydropower, the work here is focused on developing a detailed computational fluid dynamics (CFD) model of the WCFT. The computational model produced as part of this work was developed in the commercial software ANSYS Fluent. This CFD model solves the incompressible Navier-Stokes equations in their three-dimensional, unsteady form including the effects of turbulence, using detailed numerical routines. The multiphase fluid flow of air and liquid water in the turbine is simulated using a volume of fluid technique (VOF). A very detailed geometric representation of the WCFT turbine is imported into ANSYS from the computer aided design software SOLIDWORKS. Using commercial software made the development of this detailed CFD model possible within a Master’s thesis time frame.
Coupled to the computational work done here, is some experimental work. Having access to the WCFT experimental facility at Central State University was beneficial to the computational work. A good deal of knowledge about the computer modeling was gained by undertaking a small amount of experimental work. In addition, an experimental result was used to verify the average power predicted by the computational model. This comparison showed a difference of 18.1%, which is deemed reasonable given the complexities of the CFD modeling undertaken. To further verify the computational results, independence of the results on the meshing and time step utilized was demonstrated.

The primary computational results presented are plots of turbine power versus shaft rotational speed for twelve and nine bladed WCFTs. These results indicate that a nine bladed WCFT turbine performs better than the currently used twelve bladed, lab-scale WCFT. While these results are specific to the input operating conditions used in the analysis, they indicate the usefulness of the computational tool developed here. Additional results presented for the nine bladed, lab-scale WCFT are field plots of water volume fractions, many types of velocity vector plots, and field plots of the fluid pressure. Histogram plots of some of the interesting quantities are given to show the distribution of certain quantities throughout the turbine.
TABLE OF CONTENTS

CHAPTER 1 - INTRODUCTION .................................................................................. 1
  1.1. RENEWABLE ENERGY .................................................................................. 1
  1.2. FUTURE OUTLOOK FOR HYDROPOWER .................................................. 3
  1.3. POSSIBLE PROBLEMS WITH SMALL HYDROPOWER SYSTEMS .......... 4
  1.4. SMALL HYDROPOWER WORK AT CENTRAL STATE................................. 4
  1.5. OBJECTIVES OF THIS WORK .................................................................... 5
      1.5.1. General Objectives ............................................................................. 5
      1.5.2. Specific Objectives ........................................................................... 5

CHAPTER 2 - STATE OF HYDROPOWER AROUND THE WORLD.................... 8
  2.1. HYDROPOWER ......................................................................................... 8
  2.2. WORLD SCENARIO .................................................................................. 9
  2.3. UNITED STATES SCENARIO ................................................................... 11
  2.4. SMALL HYDROPOWER .......................................................................... 12
      2.4.1. World Scenario ................................................................................. 13
      2.4.2. United States Scenario .................................................................... 14

CHAPTER 3 - HYDROPOWER TURBINES ............................................................. 16
  3.1. TYPES OF HYDROPOWER TURBINES .................................................... 16
  3.2. CROSS FLOW TURBINES ....................................................................... 17
      3.2.1. Introduction to Cross Flow Turbines ................................................... 17
      3.2.2. Brief History and Development ......................................................... 18
      3.2.3. Advantages of Cross Flow Turbines .................................................. 18
  3.3. WILLIAMS CROSS FLOW TURBINE ...................................................... 19
      3.3.1. Characteristics of WCFT .................................................................. 19
      3.3.2. Advantages of WCFT ....................................................................... 19
  3.4. MODELING OF HYDROPOWER TURBINES .......................................... 20
      3.4.1. Other Hydropower Turbines .............................................................. 20
      3.4.2. Cross Flow Hydropower Turbines ..................................................... 21
CHAPTER 4 - EXPERIMENTAL WORK ON WILLIAMS CROSS FLOW TURBINE

4.1. WATER CHANNEL

4.2. LAB-SCALE MODEL OF WCFT

4.3. DIAGNOSTIC EQUIPMENT

4.4. DATA REDUCTION PROCEDURE

   4.4.1. Determination of Belt Tension
   4.4.2. Determination of Turbine and Generator Power
   4.4.3. Flowrate Measurement

4.5. EXPERIMENTAL DATA TAKEN AS PART OF THIS WORK

CHAPTER 5 - ANALYSIS TECHNIQUE

5.1. GEOMETRY

5.2. MESHING

   5.2.1. Mesh Method
   5.2.2 Near Wall Meshing

5.3. GOVERNING DIFFERENTIAL EQUATIONS

   5.3.1. Governing Equations Using the Volume of Fluid (VOF) Technique
   5.3.2. Turbulence Model

      5.3.2.1. Governing Differential Equations
      5.3.2.2. Effective Diffusivity Terms
      5.3.2.3. Production Terms
      5.3.2.4. Dissipation Terms

5.4. NUMERICAL TECHNIQUE

   5.4.1. Velocity and Pressure Field Solver
   5.4.3 Volume Fraction Solver

      5.4.3.1. Interface Modeling
   5.4.5. Turbulence Solver

CHAPTER 6 - COMPUTATIONAL RESULTS

6.1. GRID AND TIME STEP INDEPENDENCE STUDIES

   6.1.1. Grid Independence Study
   6.1.2. Time Step Independence Study

6.2. COMPUTER MODEL VALIDATION

6.3. FIELD RESULTS
6.3.1. Volume Fractions .................................................................................................................. 61
6.3.2. Velocities................................................................................................................................ 62
6.5. PRESSURE .................................................................................................................................. 67
6.3. POWER VERSUS ROTATIONAL SPEED ................................................................................... 70
   6.6.1. Twelve Bladed Turbine ....................................................................................................... 70
   6.6.2. Nine Bladed Turbine ........................................................................................................... 71
   6.6.3. Comparison of Twelve Bladed Turbine to a Nine Bladed Turbine ....................................... 72
CHAPTER 7 - CONCLUSIONS AND RECOMMENDATIONS .................................................................. 74
   7.1. CONCLUSIONS ....................................................................................................................... 74
   7.2. RECOMMENDATIONS ............................................................................................................. 77
REFERENCES ..................................................................................................................................... 79
LIST OF FIGURES

Figure 1: Schematic diagram of hydropower system. [10] ................................................................. 9
Figure 2: Hydropower production by country. [9] ............................................................................. 10
Figure 3: Hydropower capacity by region. [9] .................................................................................. 11
Figure 4: Small hydropower installed capacity by region. [13] ......................................................... 13
Figure 5: Number of small hydropower plants installations by year. [14] ....................................... 14
Figure 6: Non-powered dams in the United States with power potential more than 1 MW. [15] .... 15
Figure 7: Classifications of hydro-turbines [16] The red boxes indicate low head turbines and those turbines without a red box are high head turbines. ................................................... 17
Figure 8: Water channel in Central State’s International Center for Water Resource Management lab. ... 24
Figure 9: Lab model of WCFT and associated equipment ................................................................. 25
Figure 10: Experimental setup with water flowing. ......................................................................... 26
Figure 11: Disassembled WCFT ........................................................................................................ 28
Figure 12: Turbine runner after manufacturing ............................................................................... 28
Figure 13: Stroboscope used in measuring rotational speed of turbine. .......................................... 30
Figure 14: Digital belt frequency meter arrangement for belt tension measurement ...................... 30
Figure 15: Discharge at dam. ............................................................................................................. 33
Figure 16: SolidWorks drawing of WCFT. ......................................................................................... 38
Figure 17: WCFT mesh. ...................................................................................................................... 40
Figure 18: Inflation cell layers near blade walls. .............................................................................. 42
Figure 19: Inflation cell layers near stationary wall ......................................................................... 42
Figure 20: Flowchart of pressure based solution method. [38] ......................................................... 50
Figure 21: Actual interface (left) and interface from compressive formulation (right). ..................... 51
Figure 22: Grid independence study results. ...................................................................................... 56
Figure 23: Time-step independence study results. .......................................................................... 58
Figure 24: Instantaneous output power for base case situation. ..................................................... 60
Figure 25: Mass imbalance histogram. ............................................................................................. 61
Figure 26: Contours of water volume fraction. The air volume fraction is one minus the water volume fraction ......................................................................................................................... 62
Figure 27: Velocity vectors colored by velocity magnitude where the velocity magnitudes are given in m/s. ........................................................................................................................................ 64
Figure 28: Velocity vectors colored by density where the densities are given in kg/m3 .................. 65
Figure 29: Velocity histogram. ......................................................................................................... 65
Figure 30: Velocity vectors colored by x-direction magnitude where the velocity magnitudes are given in m/s. ..................................................................................................................................... 66
Figure 31: Velocity vectors colored by y-direction magnitude where the velocity magnitudes are given in m/s. ..................................................................................................................................... 67
Figure 32: Contours of absolute pressure .......................................................................................... 68
Figure 33: Contours of absolute pressure above atmospheric pressure. .......................................... 69
Figure 34: Pressure histogram. ......................................................................................................... 69
Figure 35: Turbine power as a function of rotational speed for a twelve bladed turbine. ............... 71
Figure 36: Turbine power as a function of rotational speed for a nine bladed turbine. ..........................72
Figure 37: Turbine power comparison between twelve and nine bladed WCFTs. .................................73
LIST OF TABLES

Table 1: Turbine flowrate calculation for different channel water flow rates. .................................. 35
Table 2: Generator and turbine powers as a function of turbine flowrates. ....................................... 35
Table 3: Base case conditions from experimental measurements. ...................................................... 54
Table 4: Power as a function of the mesh used. .................................................................................... 56
# NOMENCLATURE

List of Abbreviations

<table>
<thead>
<tr>
<th>acronym</th>
<th>full form</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSYS</td>
<td>ANalysis SYStems</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFT</td>
<td>Cross Flow Turbine</td>
</tr>
<tr>
<td>DC</td>
<td>Direct Current</td>
</tr>
<tr>
<td>DHE</td>
<td>Dayton Hydro Electric</td>
</tr>
<tr>
<td>EPRI</td>
<td>Electric Power and Research Institute</td>
</tr>
<tr>
<td>FF</td>
<td>Fluid Flow</td>
</tr>
<tr>
<td>GW</td>
<td>Giga Watts</td>
</tr>
<tr>
<td>ICWRM</td>
<td>International Center for Water Resource Management</td>
</tr>
<tr>
<td>IEA</td>
<td>International Energy Agency</td>
</tr>
<tr>
<td>kWh</td>
<td>Kilo Watt Hours</td>
</tr>
<tr>
<td>DOE</td>
<td>Department of Energy</td>
</tr>
<tr>
<td>MHP</td>
<td>Micro Hydro Power</td>
</tr>
<tr>
<td>MTOE</td>
<td>Millions of Tons of Oil Equivalent</td>
</tr>
<tr>
<td>MW</td>
<td>Mega Watt</td>
</tr>
<tr>
<td>NSD</td>
<td>New Stream-reach Development</td>
</tr>
<tr>
<td>ORNL</td>
<td>Oak Ridge National Laboratory</td>
</tr>
<tr>
<td>PRESTO</td>
<td>Pressure Staggering Option</td>
</tr>
<tr>
<td>RNG</td>
<td>Re Normalization Group</td>
</tr>
<tr>
<td>rpm</td>
<td>Revolutions per Minute</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>TWh</td>
<td>Terra Watt Hour</td>
</tr>
<tr>
<td>USA</td>
<td>United States of America</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume of Fluid</td>
</tr>
<tr>
<td>WCFT</td>
<td>Williams Cross Flow Turbine</td>
</tr>
</tbody>
</table>
### Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Cross sectional area of channel (m²)</td>
</tr>
<tr>
<td>A&lt;sub&gt;p&lt;/sub&gt;</td>
<td>Cross sectional area of pipe (m²)</td>
</tr>
<tr>
<td>A&lt;sub&gt;t&lt;/sub&gt;</td>
<td>Cross sectional area of orifice (m²)</td>
</tr>
<tr>
<td>B</td>
<td>Width of dam (meter)</td>
</tr>
<tr>
<td>C&lt;sub&gt;f&lt;/sub&gt;</td>
<td>Skin friction coefficient</td>
</tr>
<tr>
<td>D&lt;sub&gt;ω&lt;/sub&gt;</td>
<td>Diffusion of specific dissipation rate</td>
</tr>
<tr>
<td>F</td>
<td>Source term in continuity equation</td>
</tr>
<tr>
<td>f</td>
<td>Natural frequency of belt (Hz)</td>
</tr>
<tr>
<td>G&lt;sub&gt;k&lt;/sub&gt;</td>
<td>Production of turbulent kinetic energy due to mean velocity gradient</td>
</tr>
<tr>
<td>G&lt;sub&gt;ω&lt;/sub&gt;</td>
<td>Generation of specific dissipation rate due to mean velocity gradient</td>
</tr>
<tr>
<td>H</td>
<td>Height of water above the crest of the dam (meter)</td>
</tr>
<tr>
<td>h</td>
<td>Head of water flow</td>
</tr>
<tr>
<td>h&lt;sub&gt;1&lt;/sub&gt;</td>
<td>First manometer height (mm)</td>
</tr>
<tr>
<td>h&lt;sub&gt;2&lt;/sub&gt;</td>
<td>Second manometer height (mm)</td>
</tr>
<tr>
<td>Δh</td>
<td>Manometer differential height (mm)</td>
</tr>
<tr>
<td>I</td>
<td>Current Flow (Ampere)</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>L</td>
<td>Hydraulic diameter (m)</td>
</tr>
<tr>
<td>L&lt;sub&gt;belt&lt;/sub&gt;</td>
<td>Span length of belt configuration (meter)</td>
</tr>
<tr>
<td>m&lt;sub&gt;aw&lt;/sub&gt;</td>
<td>Mass transfer from phase air to phase water (Kg/s)</td>
</tr>
<tr>
<td>m&lt;sub&gt;wa&lt;/sub&gt;</td>
<td>Mass transfer from phase water to phase air (Kg/s)</td>
</tr>
<tr>
<td>N</td>
<td>RPM of turbine</td>
</tr>
<tr>
<td>n</td>
<td>Number of mass transfer mechanisms</td>
</tr>
<tr>
<td>P</td>
<td>Wetted perimeter of channel (meter)</td>
</tr>
<tr>
<td>p</td>
<td>Pressure (Pascal)</td>
</tr>
<tr>
<td>Q</td>
<td>Total flowrate in the system (m³/s)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>r</td>
<td>Radius of belt (meter)</td>
</tr>
<tr>
<td>S</td>
<td>Strain rate magnitude</td>
</tr>
</tbody>
</table>
\( S_k \) User defined source term for turbulent kinetic energy
\( S_\omega \) User defined source term for specific dissipation rate
\( S_{\alpha_w} \) Mass generation source (Kg/s)
\( T \) Torque (Newton-meter)
\( U \) Free Stream velocity
\( U_t \) Frictional Velocity (m/s)
u velocity component in different directions
\( V \) Voltage in the system (Volts)
v Shared velocity field (m/s)
\( Y_k \) Dissipation of turbulent kinetic energy because of turbulence
\( Y_\omega \) Dissipation of specific dissipation rate because of turbulence
\( y \) First height of inflation (meter)

**Greek Symbols**
\( \alpha_w \) Volume Fraction of water
\( \nabla \varphi_d \) Value of donor cell VOF gradient
\( \nabla \phi \) Cell center gradient value in an upstream cell
\( \mu \) Molecular viscosity (Stokes)
\( \mu_k \) Kinematic viscosity of water (Stokes)
\( \mu_t \) Turbulent viscosity
\( \rho \) Density (Kg/m\(^3\))
\( \sigma_k \) Turbulent Prandtl number for kinetic energy
\( \sigma_\omega \) Turbulent Prandtl number for specific dissipation rate
\( \tau_w \) Wall shear stress \( \left( \frac{kg}{m \cdot s^2} \right) \)
\( \tau_\omega \) Effective diffusivity of specific dissipation rate
\( \tau_k \) Effective diffusivity of turbulent kinetic energy
\( \phi \) Cell center scalar value in upstream cell
\( \varphi_f \) Value of face VOF
\( \varphi_d \) Value of donor cell VOF
\( \omega \) Specific dissipation rate
ACKNOWLEDGEMENT

I would like to present plethora of thanks to Prof. Dr. James Menart, the supervisor of this research and director of Renewable and Clean Energy Engineering program at Wright State University, for his initial support, unwavering encouragement and incessant expertise guidance throughout the project, without which this work would not have been possible.

My sincere gratitude are due to Prof. Dr. Subramania I. Sritharan, Associate Director for research, Land-Grant Program and former dean of College of Science and Engineering, Central State University, for his special guidance and motivating remarks that helped me to complete the work successfully. His overwhelming supports and encouragements will always be remembered.

Special thanks are due to William Fred Jr., patent holder of WCFT and founder of Dayton Hydroelectric Limited, for granting the permission to do a research on his technology, being the transportation facilitator for me from Wright State University to Central State University and his assistance in understanding about the turbine during the initial phase of research. I would also like to place on record the contribution of Dr. Krishnakumar V. Nedunuri, Department chair of Water Resource and Management, Central State University, for his co-operation in accessing the water resource lab where the WCFT is located.

I would like to acknowledge the contribution of Ohio Super Computer (OSC) and its technical support team for helping me to access and use OSC system without which the work could not have been completed in such short period of time. I would also like to remember Mike Vanhorn, Senior Computer Systems Administrator at Wright State, and all the people in the technical support team of ANSYS Fluent for helping in the technical part of ANSYS installation.

I humbly express my deep gratitude to my loving grandparents, parents, brothers and all my friends for their help and cooperation throughout the project.
1.1. RENEWABLE ENERGY

The International Energy Agency (IEA) defines renewable energy as “energy derived from natural processes that are replenished at a faster rate than they are consumed.”[1] A wide variety of energy sources are included in renewable energy, such as energy harnessed from flowing water, energy harvested from moving air (wind energy), energy harvested from the sun (solar energy) and energy harvested from many other natural forces. The ultimate sources of all types of renewable energy are the sun, the moon, and the earth’s rotation. The most widely used renewable energy today is hydropower and it is felt that this source of renewable energy needs to be tapped further.

The United States is the country with highest per capita energy consumption in the world; but, it has not been able to maintain the same position with regard to use of renewable energy. However, the trend in the use of renewable energy sources is increasing exponentially over the last few years. Diagnosing the negative effects of carbon emission on the atmosphere, investment in renewable energy is increasing, as a result more and more renewable energy resources are being developed and exploited. According to the United States Department of Energy (DOE), electricity generation capacity of renewable sources increased from 9.6 percent to 15.5 percent from 2004 to 2014.[2]
According to a report from the Institute of Energy Research, for the first ten months of 2016, 13 percent of total energy consumption in the United States is from renewable sources.[3] A prominent source of renewable energy is hydropower which occupies an overwhelming 46 percent of the total renewable energy used. The total amount of hydropower energy used in the first 10 months of 2016 is 2.091 quadrillion Btu which is equivalent to $6.50 \times 10^{11}$ KWH. Almost all of this hydropower electricity comes from large hydropower plants located across the country. Some of the large hydropower plants located in the United States are the Grand Coulee dam (6,809 MW), the Bath County PSP dam (3,003 MW), and the Chief Joseph dam (2,620 MW). The hydropower plants producing power greater than 50 MW numbers 279 throughout the United States, while the number grows to 1310 if plants greater than 1 MW capacity are considered.[4]

Although a large chunk of renewables come from big hydropower plants, there is also a tremendous hydropower potential from small hydropower plants across the country. These low head, small dam hydropower sites are often referred to as run-of-the-river hydropower. There are more than 80,000 small dams[5] throughout the United States which were constructed in the 19th and 20th centuries for irrigation, navigation and flood control purposes. None of these dams produce electricity at this time. The DOE did an assessment in 2014, called the New Stream-reach Development (NSD) assessment, in order to find the generation capacity from these untapped dams. According to the NSD assessment, the total untapped power potential in the United States is more than 65.5 GW, with yearly energy generation that could be 347 TWh. This is approximately 128 % of the average 2002 through 2011 net annual electrical generation from existing hydropower plants [5]. The Ohio River, Mississippi River, Alabama River, and Arkansas River, as well as their major tributaries, are the main locations in which such dams are present and hydroelectric energy production could be added to them. The top prospective states with small hydropower potential in decreasing order are Alaska, Idaho, California, Colorado, Kansas and Missouri.

A more remarkable fact is the monetary cost of dam construction has already been incurred. Reservoir construction cost is one of the major expenses in the development of a hydroelectric power system. On average, it accounts for twenty six percent of the total cost for the construction of a large hydropower system and eighteen to fifty percent of the total cost of a small hydropower system; this varies based on site conditions.[6] For this reason, the levelized cost of electricity that
could be produced with small hydropower systems decreases significantly. Additionally, nature has to bear no further environmental impacts of dam construction. So, adding a mechanism to pull electric power from these dams can be achieved at lower capital costs, lower risk, less environmental damage, and smaller time frames compared to developing a brand-new dam. Clearly, utilizing unused dams to produce electric energy could play a vital role in generating eco-friendly energy and should not be underestimated. The purpose of this research is to study a water turbine specifically designed to extract energy from such untapped, low head dams.

Based on the head available, water turbines are classified into distinct categories. These categories of turbines are high, medium and low head turbines. High head turbines are the turbines used for heads of 250 meters and above, whereas medium turbines are used for heads from 40 to 250 meters and low head turbines are those used for heads less than 40 meters. Generally, reaction type turbines are used in low head dams and impulse turbines are used in high head dams. Some of the turbines used in high and medium head plants are Pelton, Turgo, and Francis; whereas, turbines used in low head plants are Kaplan, propeller, screw, hydro-kinetic, and crossflow. The type of turbine for low head, untapped dams being focused on in this thesis work, fall under the category of crossflow turbines.

1.2. OUTLOOK FOR HYDROPOWER

The NSD assessment of hydropower potential in United States undertaken in 2014 has unveiled new possibilities by which hydropower potential capacity might increase by 65.5 GW upon implementing small hydropower plants. These findings could be an important milestone in hydropower development. A decreasing production trend of hydropower might be reversed and start to ascend once again. Consumption of renewables could increase as well. Thousands of renewable jobs might be opened, and hydropower companies could boom.

Meanwhile, on the 9th of August 2013, president Obama signed The Hydropower Regulatory Efficiency Act and the Bureau of Reclamation Small Conduit Hydropower Development and Rural Jobs Act in order to boost small hydropower projects. These are indications that small hydropower has a wide scope and electricity production coming from renewables will increase exponentially in upcoming years.
1.3. POSSIBLE PROBLEMS WITH SMALL HYDROPOWER SYSTEMS

Although run-of-the-river hydropower looks prolific and fruitful it has some limitations. The DOE has completed an assessment of non-powered dams nationwide, but they have not considered other factors important in hydropower electricity generation, like hydrology, conduit opportunities, canals, and pipelines. All the sites included in the NSD study might not be suitable for electricity generation because of the soil structure and land formation. Depending on the location of the site, construction of canals and pipelines might not be feasible.

Another prominent issue is grid connection. Once electricity is generated, it must be connected to the electric grid for power distribution to consumers. The DOE report does not mention the accessibility to national or standard transmission grids. Currently, the cost and technical requirements to connect a particular hydropower station to a local or national grid is uncertain. These aspects could be time consuming and expensive.

Commercial production of hydropower energy from such types of dams has only been minimal in any part of the world. In other words, these power units are not common. There are no standard design procedures similar to large hydropower plants. The equipment for use in such applications need research and development. It is difficult to get investors to take on the risks of developing new equipment for such applications.

Finally, developers may encounter a lengthy permitting process for run-of-the river hydropower plants and the development of small hydropower can be hindered by state and local regulatory guidelines. Such issues need to be addressed.

1.4. SMALL HYDROPOWER WORK AT CENTRAL STATE

Aiming to extract power from these untapped and low head dams, a special type of turbine has been designed and a lab-scale model built and installed in the Water Resource Lab of Central State University (CSU). This special turbine is designed for low head run-of-the-river dam sites and has been named the Williams cross flow turbine (WCFT). Some experimental tests have already been done and others are undertaken as part of this thesis work using the installed lab-scale WCFT. Realizing that a CFD (computational fluid dynamics) study would help to better understand the operation of this cross-flow turbine, a study using the CFD simulation software ANSYS Fluent was begun in 2015. This thesis work is that CFD study and this is the first time the
Central State, WCFT has been modeled using CFD. In addition, the Central State WCFT has been scaled for use in the Great Miami River, located in Hamilton, Ohio. The scaled turbine is being manufactured under the sponsorship of Parker Hannifin, a leading company in manufacturing motion and control devices. The CFD study presented in this thesis does not look at the full-scale turbine, but only studies the lab-scale model and possible variants. Study of the full-scale model should be undertaken by a future graduate student.

1.5. OBJECTIVES OF THIS WORK

1.5.1. General Objectives

The main objective of this work is to develop a computation model of the newly designed WCFT. To achieve this goal the commercial CFD software ANSYS Fluent is used. A modeling tool such as this can be used to do many design iterations much quicker than can be done experimentally; thus spurring the development of the WCFT, and therefore spurring the development of small hydropower installations. While the major portion of the work done for this thesis was developing this CFD tool, a second goal is utilizing this tool to study the operation of the William’s cross flow turbine. In addition, it was hoped that this study will provide one performance improvement recommendation for the lab-scale WCFT. While the WCFT operational study is relatively small, and one performance recommendation is not numerous, these results are meant to demonstrate the usefulness of the CFD computational tool developed as part of this work.

A second general objective was performing some experimental work on the Williams cross flow turbine located in the water tunnel in the Central State Water Resource Lab. While the focus of this research was computation, there were two reasons for adding an experimental portion to this thesis. The first of these is the learning that can be gained by working with real equipment. This experience was invaluable to the computational modeling performed here. The second reason for doing the experimental work was to obtain measured results to which the computational results could be compared.

1.5.2. Specific Objectives

Of course, the first specific objective is to obtain a working CFD model of the William’s cross flow turbine. As mentioned above this was done by using the CFD software ANSYS Fluent.
ANSYS Fluent [8] is a comprehensive CFD software tool that is available for purchase. The flow field in a cross-flow turbine is extremely complex and requires a robust CFD program. ANSYS Fluent is such a CFD software. Just because commercial software was used to perform this task, development of the CFD model specific to the Williams cross flow turbine still took a good deal of time and effort. Many iterations of program arrangements were undertaken in performing this task.

While undertaking the specific objective of programing a working CFD model of the Central State cross flow turbine, experimental work was being undertaken in the Water Resources Lab at Central State University. Understanding of the Williams cross flow turbine was gained during this process. In addition, experimental results that could be used to compare to the computational results were obtained.

With all numerical models, a discretization convergence study is valuable. In this work, convergence studies on the size of the grid and the size of the time step are undertaken. This work is focused on the steady state operation of the Williams cross flow turbine, but the complexity of the flow in the turbine does not produce an unchanging flow field with time, even in steady state operation. The rotating blades of the turbine are the main cause of the flow field fluctuations at steady state. On average the flow is unchanging, but there are always flow fluctuations present. These flow fluctuations are captured by the time dependent analysis performed in this work. Average output quantities are obtained by averaging over a range of time once the computations have reached steady state.

To verify the accuracy of the CFD model developed, an average power result was compared to an experimentally obtained result. The comparisons performed in this thesis are focused on output power, because this is the output performance parameter delivered by the experimental facility. It would have been nice to have velocity measurements throughout the turbine, but these are difficult measurements to make in a cross-flow turbine and are not available at the present time.

A fifth specific goal of this work is to produce turbine power versus turbine blade rotational speed. In this thesis, these curves will be referred to as power curves. For a given water mass flow through the turbine at a given head, the turbine output power changes as the rotational speed of the blades changes. Thus, to find the best performance of a given turbine design, in a given flow
situation, several blade rotational speeds have to be simulated. This is done in this work so that more conclusive turbine performance information is produced.

As mentioned above, obtaining velocity field information in the turbine is desirable. Velocity magnitude and velocity direction information is obtained as part of this work. Other field information obtained are the volume fractions of air or water at specific locations in the turbine and pressure contours. To obtain such information experimentally would be extremely difficult.

The seventh and last specific objective of this work is to produce power curves for a 9 bladed Williams cross flow turbine and a 12 bladed Williams cross flow turbine. At the start of this project the question arose as to how many blades should be used in the turbine. Some results have been generated in this thesis that begins to answer this question.

1.6. OUTLINE OF THESIS

The next chapter of this report will delve deeper into hydropower status around the world. The current state of large scale and small-scale hydropower across the globe is described. Chapter three describes the types of water turbines used in different hydropower plants. This includes turbines used in large to small hydropower stations, but mostly focuses on cross flow turbines (CFT) and in particular the WCFT, which is the turbine modeled in this research work. Chapter 4 presents the experimental facility used to make measurements on the WCFT developed at Central State University. The data reduction procedure and the final measured results done as part of this work are also shown. In Chapter 5 the details of the computational model used to simulate the WCFT are given. The construction of the turbine geometry in solid works, the meshing, the governing differential equations, and the numerical technique used to solve these governing differential equations are given here. Chapter 6 presents the results obtained from the computational analysis. These results include grid and time step independence studies, a comparison to an experimentally measured power, power curves, a performance comparison between a 9 and 12 bladed Williams cross flow turbine, and presentation of some field quantities. Finally, Chapter 7 ends this thesis with some conclusions and recommendations on the CFD model developed and its use for advancing the state of the art in WCFT and for advancing run-of-the-river hydropower installations.
CHAPTER 2

STATE OF HYDROPOWER AROUND THE WORLD

2.1. HYDROPOWER

Hydropower is the energy source that comes from the kinetic or potential energy stored in water. Thus, fast moving water is a source of hydrokinetic energy that can be tapped for electricity production or some other useful purpose. In addition, water stored high above a lower location is a source of hydro potential energy. The ultimate source of hydropower is solar energy. The sun evaporates water from the sea or other body of water and the wind drives the moisture overland. The moisture turns into a cloud after condensation and eventually falls to the earth’s surface as rain or snow. Gravity attracts water from a high to low altitude and this flowing water is harnessed to produce an energy form useful to humankind.

Hydropower is not only the most widely used renewable energy in the United States, but also world. In 2016, it supplied seventy-one percent of all renewable energy and sixteen percent of world’s total electric energy generated.\(^9\) In the United States, the number is 6.8% and 44.15% for total and renewable electricity share respectively.

There are three main systems of a typical hydropower installation, civil, mechanical, and electrical. The civil system includes the reservoir, dam, spillway and intake (see Figure 1). The mechanical system includes the penstock, turbine, and tailrace; while generators, transmission and distribution lines, and transformers are included in the electrical system.
2.2. WORLD SCENARIO

Harnessing energy from water started as long ago as modern human civilization existed. There is no unanimity among historians about where and when the first hydraulic technology emerged; however, it has been found that the Greeks and Romans used water energy to grind wheat to flour. Coming from the ancient era to today’s date, there has been tremendous advancement in the methodology used to capture power from flowing water.

Figure 2 displays the current hydropower scenario around the world. Today, China is on the top of the list in generating hydroelectric power followed by Brazil, Canada, United States, and Russia. Other countries producing hydroelectric energy are Norway, India, Japan, Venezuela and Sweden, as shown in Figure 2. China alone produces 96.9 MTOE (Millions of Tons of Oil Equivalent) of hydroelectric power which is more than the combined production of Brazil, Canada and the United States. Statistics indicate that China has exploited about sixty five percent of its
economically feasible hydropower resources at this time \cite{9}, while the United States has utilized more than seventy percent.

According to a recent World Bank report \cite{11} Albania and Paraguay produce all of their electricity from hydropower. Tajikistan, Nepal, Zambia, Democratic Republic of Congo, Mozambique, Norway, Ethiopia and Namibia are countries which produce more than 95\% of their electricity from hydropower.

![Figure 2: Hydropower production by country. \cite{9}](image)

Although, generating hydropower is site specific, and not available in every corner of world, it is one of the most dispersed energy sources. The most important parameter in generating electricity is an elevation change along the flow path of the water. This makes geographical locations with mountains and hills more suitable to tap hydropower. According to geographical location, as shown in Figure 3, East Asia has the highest hydropower generating capacity, which is about one third of the world’s capacity. The sequence is followed by Europe, North America,
and South America. Africa and the Middle East are the regions with the lowest hydropower potential. In East and South Asia, the Eurasian plate is moving against the Pacific and Indian plates to produce high land structures. Similarly, the South American plate and Nazca plate in the west of South America and the Pacific and North American plates in the west of North America move against each other forming suitable topographical structures to harness hydroelectricity.

Figure 3: Hydropower capacity by region. [9]

2.3. UNITED STATES SCENARIO

Although, the United States was the first country to produce energy from flowing water, generation started on the 30th of September, 1882, widespread commercial production started in the early twentieth century after the establishment of the Bureau of Reclamation in 1902 and congress authorized the US Corps of Engineers to be involved in building hydropower dams in 1920. Today, the Bureau of Reclamation is the second largest producer of hydropower and US
Corps of Engineers operates 75 power plants across the country. The United States produces approximately 45 percent of its total renewable energy from hydropower. The current capacity of hydropower in United States is 101 GW, 79.6 GW from hydropower generation and 21.4 GW from pumped storage hydropower facilities.\textsuperscript{[12]}

During the economic growth time in the United States, from the early to mid-twentieth century, significant development of large hydropower plants took place. The Grand Coulee dam (6809 MW, 1942), Chief Joseph Dam (2620MW, 1958), John Day Dam (2160MW, 1941) and Hoover Dam (2080MW, 1936) are some of dams that epitomize this golden era of dam construction in the United States. After the mid-twentieth century, when the environmental consequences of these large dams were characterized and the most efficient sites to trap water energy were utilized, the development of hydropower plants halted and the total electrical generation capacity from water remained almost constant.

Today, the total number of hydropower producing sites of more than one megawatt are 1310. Out of these, 184 sites can produce more electricity than 100 MW. Most of these large hydropower sites are located in the states of California, Alaska, Washington, Oregon, and Colorado. The topographical geography of these states resembles each other. California has forty-one mountains exceeding ten-thousand feet, Washington and Oregon have five and six respectively, while Colorado has more than fifty, and Alaska has sixty such types.

\textbf{2.4. SMALL HYDROPOWER}

Small hydro plants are relatively small in terms of power output and size, but no universal definition for the capacity of small hydropower systems has been set at this time. Small hydropower is divided into various categories depending on the production capacity; pico, micro, and mini are its classifications in increasing capacity size. The capacity ranges for pico, micro, and mini are different from country to country. The DOE in the United States defines small hydropower as projects that can generate less than 10 MW of electricity, whereas the Indian government uses a limit of 25 MW.
2.4.1. World Scenario

Small hydropower replicates the distribution of large hydropower around the world. According to a World Bank report,[13] the total installed capacity of small hydropower in 2016 is 78 GW; this is 36% of the total potential capacity. As depicted in Figure 4, Asia is the region with the highest installed capacity, which is approximately two thirds of the total world capacity followed by Europe, America, Africa and Oceania (Australia and New Zealand). Asia and Europe have developed more than 40% of their total potential small hydropower plants; whereas, the Americas (both North and South) have developed only 17%. On the other hand, Africa has only developed 4% of its total potential capacity.

Similarly, on a country basis, China is a superpower in generating energy from small hydropower. (China defines small hydropower as being below 10 MW.) Its capacity is four-fold the combined capacity of the other four largest small hydropower countries; namely Italy, Japan, Norway, and the United States.

Figure 4: Small hydropower installed capacity by region.[13]
2.4.2. United States Scenario

During the initial stage of development of hydropower in the United States, mostly larger hydropower units were developed. Figure 5 describes the development of small hydropower plants in the United States. This figure shows that the number of small hydropower plants was only a few until the 1970s. The number of small hydropower units increased significantly in the 1970s after the implementation of the Wild and Scenic Rivers Act in 1968, the National Environmental Policy Act in 1969, the Fish and Wildlife Coordination Act in 1974, and the Public Utility Regulatory Policies Act in 1978. These acts, launched under the presidency of Richard Nixon, regulate the construction of large hydropower plants ensuring environmental and natural resource protection, including fish and wildlife. The Public Utility Regulatory Policies Act in 1978 encourages construction of small hydropower by exempting certain small plants from federal licensing requirements. The last two decades of the twentieth century is regarded as the golden era in development of small hydropower. More than 700 different sites were installed during this period.

![Figure 5: Number of small hydropower plants installations by year.][14]

In the 21\textsuperscript{st} century, the growth of small hydropower has decreased, and the installed capacity has almost remained constant. Recently the DOE tasked the Oak Ridge National Laboratory (ORNL) to evaluate the New Stream-reach Development (NSD) potential for more
than three million United States streams. This work was carried out from 2011 to 2013. After the assessment was completed across the entire United States, numerous new potential sites were discovered. The total undiscovered capacity was found to be 84.7 GW and annual power generation was estimated to be 460 TWh/year. Excluding the areas protected by federal legislation from new development of hydropower, the total capacity falls to 65.5 GW, more than 80% of the current hydropower generation capacity in the United States. Figure 6 shows the distribution of untapped dams across the United States with their capacity. The Pacific Northwest region is found to have the highest potential capacity followed by the Missouri region and the California region. While comparing the states, the highest potential states are Oregon, Washington, Idaho, California, Colorado, Alaska, Arizona, and Pennsylvania.

Figure 6: Non-powered dams in the United States with power potential more than 1 MW.^[15]
CHAPTER 3
HYDROPOWER TURBINES

3.1. TYPES OF HYDROPOWER TURBINES

Different hydro-turbines are used in different hydropower facilities. The main reasons for suitability of several types of turbines at different locations is the head of the water source available. Some turbines are more efficient at high heads, while others are more efficient at lower heads. Another aspect of the water source available that effects the choice of turbine type is the flow rate. Two major classifications of hydro-turbines are reaction turbines and impulse turbines. Figure 7 shows these two basic categories and lists a number of turbines that fall into these categories.

Reaction turbines are the types of turbines mainly suitable for low or medium heads and high flow rate sites. The driving force in these turbines is the pressure difference across the runner blades. The rotor is fully immersed in the water and the turbine rotor is enclosed in a pressure casing. The Francis turbine is an example of a medium head reaction turbine, whereas the propeller/Kaplan turbine is a low head reaction turbine.

Impulse turbines are turbines which use the velocity of the water to rotate the blades. Unlike reaction turbines, these are best fitted for high head and low flowrate water resources. Widely known impulse turbines are the Pelton turbine and the Turgo turbine. Ironically, the crossflow turbine is also an impulse turbine, but it is used for low to medium head applications. In Figure 7, red boxes around the turbine name indicate that it is a low head turbine. This work is investigating the use of a cross flow turbine for a low head application.
3.2. CROSS FLOW TURBINES

3.2.1. Introduction to Cross Flow Turbines

A cross flow turbine (CFT) is a device used to convert kinetic energy, and maybe a little potential energy, of water to rotational mechanical energy of a shaft. Initially designs had water flowing through the top rotor blades to produce torque on the rotor, the water then went to the center of the rotor, and finally to the bottom rotor blades producing more torque. That is, each element of water flowed through the blades twice. This enabled the turbine rotor to extract energy from the water flow twice, increasing the efficiency of the CFT.
The CFT is one of the earliest developed water wheels in the history of water turbines. The main difference between a CFT and other conventional hydro-turbines is the flow path of the water. In other turbines, water flows in the axial or radial directions; whereas, in a CFT the flow is across the turbine rotor. Other special characteristics of the CFT include the runner orientation and the runner-shaft orientation. In CFTs, runners are not connected to a central shaft; instead they are connected to two parallel discs, one on each side of the turbine. In every other water turbine, turbine blades are connected to a central shaft. Moreover, the position of the runner shaft is always horizontal in cross flow turbines; whereas, in other turbines it could either be horizontal or vertical.

3.2.2. Brief History and Development

The CFT is an outgrowth of a scholarly exchange between Anthony Mitchell, Australian inventor, and the German entrepreneur, Fritz Ossberger. Although, initially patented in Germany by Ossberger in 1922 as a free-jet turbine (Imperial Patent No. 361593), the first CFT was eventually patented under its current name in 1933 (Imperial Patent No. 615445) after numerous refinements from the free-jet turbine. After the free-jet patent registration, production of the Ossberger CFT started commercially in Germany and, after being granted a trade license was transported overseas for commercial use. Each new location in which the CFT was used involved modifications of the design to make the turbine more suitable for that particular location’s requirements.

3.2.3. Advantages of Cross Flow Turbines

Crossflow turbines are one of the most widely used water turbines for low head applications. There are a few advantages of CFTs over other turbines which makes them widely acceptable. CFTs have a flat efficiency curve for loads ranging from 20% to 100% which makes them suitable for run-of-the-river applications where the flow varies with the season of the year. Not only do CFTs operate efficiently at different load conditions, they also can be used for a wide range of heads. They have a useful head range from 2 to 200 meters, which is the operating head range for most hydropower plants.

Moreover, CFT’s are easy to manufacture and to install. The blades are made up of parts with a single curvature which makes them easy to produce. This means local manufacturing of
cross flow turbines is easier to imitate. Additionally, since CFTs can be produced locally without rigorous design procedures and they can be built using local materials, this makes CFTs more economical compared to other hydro-turbines. Finally, because of fewer moving parts in a CFT, the maintenance frequency is lower and usually can be done on site.

3.3. WILLIAMS CROSS FLOW TURBINE

As already stated in Section 1.1 of this report, more than 80000 untapped dams can be used to generate electricity. When considering this huge future possibility, the Principal of Dayton Hydro Electric (DHE) Ltd., Mr. Fred Williams Jr., developed a special type of CFT exclusively suitable for locations where the dam is already in existence. The technology was patented by Fred Williams in the United States Patent and Trademark Office, patent number 5,882,143, on March 16, 1999 as a “low head dam hydroelectric system”. This type of turbine can directly be placed at the tail of these untapped dams with minimal or no environmental effect. A lab-scale model of this turbine has been manufactured in Xenia, Ohio, funded and sponsored by the Electric Power and Research Institute (EPRI) and installed in the hydraulics lab of the International Center for Water Resources Management (ICWRM) located at Central State University.

3.3.1. Characteristics of WCFT

WCFTs are made to use directly under the untapped dams. This eliminates the construction of any civil works which are an integral part of other small and micro hydropower plants. These structures include the dam, the intake, the canal, the forebay, the settling basin and the spillway. In addition to civil works, use of WCFTs eliminates some hydro-mechanical structures like penstocks, nozzles, and flow control valves. Moreover, the entrance of the water into the runner is either horizontal or vertical in Ossberger CFTs; whereas in WCFTs it is neither horizontal nor vertical, but rather makes an acute angle to the horizontal surface.

3.3.2. Advantages of WCFT

WCFTs have a number of advantages over other CFTs. Probably the most important of these advantages is the estimated low cost of a WCFT. WCFTs should be more economical than other CFTs due to the elimination of additional civil structures and the elimination of some hydro-
mechanical structures. The elimination of additional structures that are required with many CFTs, should tilt the economics in favor of WCFTs.

Like all hydropower electricity production, WCFTs will reduce carbon emissions and other air pollutants involved with the burning of fossil fuels. What may not be realized is by placing WCFT at sites with already built dams, construction carbon emissions are reduced. Thus, WCFTs actually do reduce carbon and other pollutant emissions compared to other hydropower turbines that require a great deal of dam construction or alteration for them to be installed. On top of lower pollutant emissions, WCFTs do not affect the downstream water flow. WCFT do not divert water in the river from its original path. Because of the special characteristic of WCFTs, the downstream ecosystem and downstream irrigation are not affected by this type of energy production.

Lastly, WCFTs are safer for aquatic life. WCFTs spin at relatively low speeds, not more than 160 rpm, which gives fish passing through them a higher chance of survival. This, along with the wide gap between blades, means small fish can travel right through the WCFT without being hurt. Fish that are larger than the blade separation will experience difficulties in passing through a WCFT unhurt.

### 3.4. MODELING OF HYDROPOWER TURBINES

#### 3.4.1. Other Hydropower Turbines

Researchers working on modeling hydropower turbines have mainly focused on design and optimization of commercially available turbines such as Pelton, Francis or Kaplan turbines in terms of shape, size, orientation, and material used in turbine manufacturing [19]. Research is also being done in the areas of turbine corrosion, cavitation in turbines, and sediment erosion of turbines. [20], [21] In addition to these topics, some research has been focused on the effect of different parameters such as pressure variation inside the runner, turbulence modeling in turbines, and grinding on the life of aquatic animals. [22]–[24]

With the aid of a k-ω SST turbulent model, an analysis using ANSYS-CFX was performed by Jošt et al. to determine the flow field of a two jet, Pelton turbine. [25] This Pelton wheel had a horizontal axis and the computed results were compared to experimental results to validate the CFD model. The analysis was done at several operating points in different operating regimes. In 2012, a CFD analysis of a Pelton turbine was done Barstad from the Norwegian University of
Science and Technology that tested a Pelton wheel for different operating conditions [26]. Another numerical approach for CFD analysis of a Francis runner was accomplished by Shukla et al. in 2011 in India. [27] They simulated the experimentally tested lab-scale turbine and predicted its behavior in actual operating conditions. Numerous efforts were done to study the cavitation problem in a Francis runner as it operates below atmospheric pressure and has a high probability of cavitation. Simulations of two phase, unsteady, cavitating turbulent flow in a Francis turbine was done by Wu et al. in 2010 using a RNG k–ε turbulence approach with three-dimensional unstructured tetrahedral mesh structures. [28] They concluded that the pressure fluctuation at small flow rates is larger than at large flow rates. Another numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation was done by Zhang and Zhang in 2012 with the standard k-ω SST turbulent model. [19] In 1997, Cada and Coutant from Tennessee concluded that increases in the pressure within a hydroelectric turbine are less of an issue than decreases in the pressure for the safety of aquatic animals. They also determined that pressure reduction to more than sixty percent of ambient, leads to cavitation and cavitation-related injury to fish. Wand et al., from Michigan State University, in 2012 investigated the axial turbine design using composite materials [29]. Torque and power at different operating conditions were calculated and analyzed for specific flow speed.

3.4.2. Cross Flow Hydropower Turbines

Various experimental and computational research has been done after the patent registration of the CFT in 1933. In 1949, Mockmore and Merryfield, Professors of civil engineering in the Oregon State system of higher education, designed a lab-scale Banki Water Turbine and obtained power and efficiency curves under different head conditions [30]. Another experiment was done by Khosrowpanah et al., members of the American Society of Civil Engineers, in 1988, by varying the number of blades, diameter of the runner, and the nozzle entry arc under different head conditions [31]. The original Banki CFT was modified and numerous experimental data was taken in order to optimize the turbine in terms of blade number, flow rate, head, and nozzle orientation. The latest experimental research on CFTs was done by Soenoko et al. from Brawijaya University, Malang, Indonesia, in 2014, entitled “Second Stage Cross Flow Turbine Performance,” which tested turbine performance under different nozzle attack angles of 30, 45, 60 and 70 degrees [32].
Apart from experimental work, several computational investigations were done to study and improve the performance of CFTs. Computational work has mainly been focused on the internal water flow behavior, blade orientation, blade angle, and discharge regulator. One of the pioneer investigations was done by Choi et al., in 2007, from Korea Maritime University, which used CFD analysis to simulate the performance of cross flow hydraulic turbines with varying blade angles [33]. Two other CFD studies that were done using ANSYS-CFX to optimize the Banki-Michell CFT were conducted by Sammartano et al., in 2013, in Italy, which optimized the number of blades and attack angle for a given head and discharge rate; [34] and the study done by Sinagra et al., in 2014, to investigate the effect of relative velocity change of the water entering the turbine with respect to the velocity of the rotating blades. [35]
CHAPTER 4
EXPERIMENTAL WORK ON WILLIAMS CROSS FLOW TURBINE

As part of this thesis work a few experiments on a Williams cross flow turbine (WCFT) were undertaken at Central State University. Central State’s International Center for Water Resource Management is well equipped to perform very low head and low flow hydropower experiments. Central State’s lab is equipped with a water channel, a small scale WCFT that fits into this water channel, and the required diagnostic tools.

4.1. WATER CHANNEL

A picture of the water channel used in these experiments is shown in Figure 8 below. This is a closed loop water channel that is 25 m long with a rectangular shaped cross section that has a height of 0.4 m and a width of 0.3 m. The system has an orange frame to support the water channel and a blue steel beam that supports the water channel and the orange framing. The base of the channel that carries the water is four feet above the floor of the laboratory. The top of the water channel is open and is in contact with the atmosphere, while the two sides and the bottom are made of transparent plastic. The flow of water can easily and distinctly be seen from the two sides which is a nice aspect of the Central State water tunnel. The return path for the water is a rectangular tank that runs along the left side of the channel as shown in Figure 8. The tank has a capacity of 200 liters and acts as water storage for the water channel. This tank is covered with plywood (as seen in Figure 8) so that people can stand on it to view the water channel from the top. The water storage
tank is connected to the water channel with pipes at both ends. This water channel is capable of producing waves, but this capability was not used for this work.

Located at the end of the water channel shown in Figure 8 is a pump that circulates water from the storage tank, through the water channel, and back through the storage tank. A flow control valve is placed near the pump, which allows the operator to set a desired flow rate. Changing the flow rate also changes the water head delivered to the turbine. The maximum head that can be achieved in this water tunnel is 0.20 meters.

![Figure 8: Water channel in Central State’s International Center for Water Resource Management lab.](image)

A picture showing the placement of the WCFT in the water channel is given in Figure 9. You cannot see the rotor of the turbine here, but the casing of the WCFT and other parts of the apparatus are visible. The larger, lower pulley shown in Figure 9 is connected to the shaft of the WCFT rotor. The smaller, upper pulley is connected to the shaft of the electric generator. The silver object to the far right in the figure is the dam. It is this obstruction that is the primary cause of a water elevation difference between the upstream side of the turbine and the downstream side.
The triangular shaped channel leaning against the dam and connected to the turbine casing directs water off the dam into the turbine. This is a type of penstock. At the downstream side of the turbine is a tail race that guides the water from the turbine back to the main flow of the water channel. The sides of the penstock and the tail race are made of clear plastic so that the water flowing in and out of the turbine can be seen. A picture that provides the reader with a visual representation of how the water flows over the dam, into and around the turbine casing, and out of the tail race is provided in Figure 10. Note the height of the water above the dam. This height is controlled by changing the water flow rate through the channel.

Figure 9: Lab model of WCFT and associated equipment
4.2. LAB-SCALE MODEL OF WCFT

A lab-scale model of a WCFT that has a diameter and width of six inches has been constructed and placed in a casing. The rotor of this WCFT is clearly shown in Figure 11 where one side of the steel casing has been removed. Figure 12 shows the rotor completely removed from the casing. In Figure 11, the left side of turbine is the inlet and the right side of the turbine is the exit. This is flipped from the as installed pictures shown in Figures 9 and 10.

Each turbine blade has two parts, a curved portion and a straight portion. The curved and straight portions of the turbine blades can clearly be seen in Figures 11 and 12. The curved part of blade is situated along the outer circumference of the rotor, while the straight part is near the center. These two parts of the blades can be separated and used alone for an experiment, if desired. The straight portion of these blades are 0.827 inches long and the curved portion covers 0.8 inches of
the radius of the rotor. The radius of curvature on these blade tips is 0.604 inches. The curvature at the end of the blades is designed to turn the water flow and convert as much of the kinetic energy of the water into rotational energy of the rotor as possible. The blades are attached to the two end walls of the rotor and not to a shaft. This connection is clearly shown in Figure 12, but the plastic end wall can also be seen in Figure 11. The shaft of the turbine extends across the entire turbine. This is difficult to see in Figure 12 because of the metal blades and light reflections off the metal blades. The picture angle in Figure 11 shows one end of the shaft. The shaft for the WCFT used in these experiments has a diameter of 0.45 inches. The shaft on one side extends outwards from the side wall by 1.80 inches so that a pulley can be attached to it and used to run an electric generator. The shaft on the other side extends 1.20 inches from its side wall.

The blades of the WCFT lab-scale model were manufactured by TDL Tool Incorporation located in Xenia, Ohio. The thickness and radius of curvature of the blades are 0.12 inches and 0.604 inches, respectively. Blades are made from aluminum sheet metal which is bent to a curve using hydraulic press. These blades are attached to the two side disks of the turbine rotor where a track has been cut to fix them in place. The blades of this lab-scale WCFT are designed and fabricated in such a way that they can be detached from the side walls easily. This makes it so the number of blades in the turbine can be altered easily. The maximum number of blades that can be installed in this lab-scale WCFT is twelve, but the number of blades actually used can be made to any number smaller than twelve. Since the position of the blades on the side walls is fixed, the gap between two consecutive blades will be different unless the number of blades is twelve, four or two.

4.3. DIAGNOSTIC EQUIPMENT

The diagnostic equipment used for this experiment includes an electric generator, light bulb array, a volt meter, a current meter, a belt tension gage, and stroboscope. These pieces of equipment are discussed below, while the process of reducing the measured quantities into the desired quantities is described in the next section.
Figure 11: Dissembled WCFT

Figure 12: Turbine runner after manufacturing
A direct current permanent magnet motor rated for a power output of 105.6 watts is used as a generator. The maximum designed rotational speed of this motor is 4200 rpm. Lesson DC motors is the manufacturing company of the motor and its model number is M111006.00. The rated armature voltage and armature current are 240 volts and 4.4 amperes, respectively.

The generator is connected to six, 0.5-watt light bulbs connected in series (see Figure 10). These light bulbs add a load to the generator. By adding a load to the generator, the rotational speed of the turbine can be changed. As shown in Figure 10, between each of the six light bulbs is a switch. These switches allow 1, 2, 3, 4, 5, or 6 light bulbs to be the load on the generator. In Figure 10 all six lights are lit by energy from the water, converted to rotational energy by the WCFT, converted to electrical energy by the generator, and converted to light and heat by the light bulbs.

To determine the output power of the turbine, the torque and rotational speed delivered by the turbine shaft are measured. The rotational speed of the turbine shaft is measured by a stroboscope (see Figure 13). The torque delivered by the turbine is determined by measuring the tension in the belt connecting the turbine to the electric generator. This tension is measured with a digital belt frequency meter. The arrangement of this belt frequency meter relative to the belt can be seen in Figure 14. The primary function of the belt frequency meter is to measure the frequency of vibration of the belt. During a measurement, the belt frequency meter is placed normally, close to the vibrating belt, and detects the frequency of belt in hertz. From this frequency the tension in the belt can be determined. With the belt tension, the torque delivered by the shaft of the turbine can be determined. The turbine power can be determined once the torque and rotational speed are known.

The electrical power output of the generator is determined by making voltage and current measurements. The voltage and current measurement is made with a single device called a Fluke multimeter. The current rating of this Fluke multimeter is 10 A while voltage range is 1000 V. The output power of the generator cannot be used to indicate the power output of the turbine because of the low efficiency of generator at the operating conditions of these experiments.

The total water flow in the system is measured using an orifice flow meter at the inlet to the flume. This orifice meter is fitted with a differential manometer which measures the flow with a manometer differential height. As the water passes through the orifice, because of change in cross
sectional area, a drop-in pressure is realized. This magnitude of this pressure drop is measured as the differential height in a manometer and this differential height is used to calculate the water flowrate through the water channel.

![Stroboscope](image1)

**Figure 13:** Stroboscope used in measuring rotational speed of turbine.

![Digital belt frequency meter arrangement](image2)

**Figure 14:** Digital belt frequency meter arrangement for belt tension measurement.

### 4.4. DATA REDUCTION PROCEDURE

The conversion of the quantities measured with the equipment described in the previous section, to the desired operating quantities of the WCFT is described below. As mentioned above, the primary quantity sought is the power output of the turbine. To get the power output of the turbine, the belt tension and rotor rotational speed need to be determined. The rotor rotational
speed is directly measured in revolutions per minute which should be converted to radians per second. This unit conversion is included in the turbine power formula. Another operating parameter of interest is the water flowrate through the turbine. The determination of this quantity is described below as well.

### 4.4.1. Determination of Belt Tension

From the measurement of the belt frequency by the digital belt frequency meter the belt tension can be determined as

\[ T = \rho L^2 f^2, \]  

where \( \rho \) is the mass per unit length of the belt and \( L \) the length of the belt between the pulleys as shown in Figure 14. For the experimental arrangement used in this work \( \rho = 0.00095 \) kg/m\(^3\) and \( L = 0.21 \) m. These numbers were obtained with a measuring tape and weight scale. To get the belt vibration frequency, three measurements were performed and the average of the three readings was used in Equation (4.1). A typical number for the vibration frequency of the belt is 70 Hz and a typical number for the belt tension is 0.20 N.

### 4.4.2. Determination of Turbine and Generator Power

The turbine power is determined from

\[ P_T = \frac{2\pi TN r_p}{60} \]  

This formula comes from the fact that the power being transmitted through the belt is

\[ Power = force \times velocity \]

The velocity of the belt is

\[ V = \omega_p \times r_p \]

where the angular speed of the turbine pulley is \( \omega_p \) and the radius is \( r_p \). If the rotational speed of the pulley is measured in revolutions per minute, the angular speed for the pulley can be written as

\[ \omega_p = \frac{2\pi N}{60} \]
where $N$ is the rotational speed in revolutions per minute. Thus, the power produced by the turbine is simply Equation (4.2). The tension in the belt, $T$, is obtained from Equation (4.1).

The power from the generator is simply determined by multiplying the measured voltage by the measured current running through the light bulb circuit. This power is noticeably less than the power from the turbine due to inefficiencies in the generator and friction losses in the transfer of power from the turbine to the generator. At the WCFT rotational speeds used in these experiments the generator is very inefficient. This does not cause any difficulties because the power from the WCFT is measured directly.

### 4.4.3. Flowrate Measurement

The formula used to calculate the total volumetric flowrate through the water channel is

$$
\dot{Q} = \frac{A_t C_d}{\sqrt{1 - \left(\frac{A_t}{A_p}\right)^2}} \sqrt{2g(h_1 - h_2)},
$$

(4.6)

which may more simply be written as

$$
\dot{Q} = C_o C_d \sqrt{\Delta h},
$$

(4.7)

where

$$
C_o = \frac{A_t}{\sqrt{1 - \left(\frac{A_t}{A_p}\right)^2}} \sqrt{2g}.
$$

(4.8)

In these equations, $A_p$ is the cross-sectional area of the pipe and $A_t$ is the area of the orifice. The quantity $(h_1 - h_2)$, which equals $\Delta h$, is the manometer differential height. The discharge coefficient $C_d$ is obtained from calculations based on sharp edged orifices. Numerical values used for the coefficients are $C_d = 0.742$ and $C_o = 0.050$. The areas in Equations (4.6) and (4.8) are $A_t = 0.00636 \text{ m}^2$ and $A_p = 0.00769 \text{ m}^2$. 


Because there is considerable leakage around the dam placed in the water tunnel, another means of determining the flow rate over the dam is required. This can be done by using equations for the flow over a weir. The volumetric flow rate over the weir (see Figure 15) is calculated using

\[
\dot{Q} = C_{d,\text{dam}} \frac{2}{3} \sqrt{2g} \times H^\frac{3}{2} \times B
\]

(4.9)

Where, \(B\) is the width of dam and \(H\) is the height of the water above the top of the dam. The discharge coefficient \(C_{d,\text{dam}}\) is obtained from approaches based on nappe profiles of flow over sharp crested weirs or a critical flow assumption at the top of a dam. With the appropriate insertion of constants and acceleration due to gravity, Equation (4.9) takes the form

\[
\dot{Q} = 2.191 B H^\frac{3}{2}.
\]

(4.10)

where standard metric units need to be used for all quantities in this equation. When this is done the volumetric flow rate is given in m\(^3\)/s.

Figure 15: Discharge at dam.
An assumption made in this work is that the flowrate into the turbine is half the flowrate over the dam. This is a reasonable assumption since the width of turbine is exactly half the width of the dam. The dam is the same width as the water channel, but does allow a significant amount of water to leak around its sides.

4.5. EXPERIMENTAL DATA TAKEN AS PART OF THIS WORK

A series of experiments were performed in Central State’s International Center for Water Resource Management Lab to investigate the characteristics of the model WCFT. The deliberate variations made during the experiment were changes in water flowrate through the tunnel and changes to the load on the turbine. Both of these changes resulted in a change in turbine rotational speed.

The experimental results documented as a part of this work are presented in Tables 1 and 2. Flow rates through the orifice and the flow over the dam are calculated using equations (4.7) and (4.10). From Table 1, it can be seen that these two flow rates are different; however, these should be the same. Reason that these two flowrates are different is total volume of water in the water channel from the orifice meter does not get passed over the weir. There is a small gap in between the weir and the side wall of the water channel on either side from which some fluid passes, making the volume flowrate over the dam smaller than that through the channel. The turbine flow is considered to be exactly half of the dam flow since the width of dam is two times of turbine model.

Six sets of experiments were done by changing the water volume flowrate. The change in flowrate was controlled by a valve present between the entrance to the water channel and the water storage tank. During these experiments, no additional load was placed on the turbine other than that imposed by the generator running with no lights on. In Table 1, the quantities H’ and H are the heights of water level above the base of the water channel and the weir respectively. This height differential is used in calculating flowrate from the dam; while the algebraic difference of h₁ and h₂ is used in calculating flowrate through the water channel. We can see in the table that with increasing manometer differential height, the water volume flowrate through the orifice and over the weir increases. This increased water flowrate over the weir increases the turbine flowrate and
this increases the rotational speed of the turbine. Ultimately, an increasing water flowrate, increases the power output of the turbine.

The primary quantity of the experimental results presented in Table 1 is the power output from the turbine. Also shown is the power output from the electric generator. The power output from the turbine is calculated using Equation (4.2), while the power from the generator is the product of voltage and current measured using the multi-meter. The generator output power obtained is far less than the power output of the turbine. In these experiments, the generator is behaving in a very inefficient manner. It is suspected that the rotational speed of the generator is much lower than its design rotational speed. The power output of the generator has no bearing on the work presented in this thesis, because the power output of the turbine has been measured independent of the generator. Design changes to the experimental apparatus can be made in the future to increase the efficiency of the electric generator.

Table 1: Turbine flowrate calculation for different channel water flow rates.

<table>
<thead>
<tr>
<th>$h_1$ (mm)</th>
<th>$h_2$ (mm)</th>
<th>$\Delta h$ (mm)</th>
<th>$H'$ (mm)</th>
<th>$H$ (mm)</th>
<th>Channel Volumetric Flowrate (L/s)</th>
<th>Dam Volumetric Flow Rate (L/s)</th>
<th>Turbine Volumetric Flowrate (L/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>684</td>
<td>654</td>
<td>30</td>
<td>404</td>
<td>47.0</td>
<td>6.91</td>
<td>6.69</td>
<td>3.35</td>
</tr>
<tr>
<td>704</td>
<td>638</td>
<td>66</td>
<td>413</td>
<td>56.0</td>
<td>10.3</td>
<td>8.71</td>
<td>4.35</td>
</tr>
<tr>
<td>776</td>
<td>593</td>
<td>183</td>
<td>433</td>
<td>76.0</td>
<td>17.1</td>
<td>13.8</td>
<td>6.88</td>
</tr>
<tr>
<td>817</td>
<td>567</td>
<td>250</td>
<td>442</td>
<td>85.0</td>
<td>20.0</td>
<td>16.3</td>
<td>8.14</td>
</tr>
<tr>
<td>840</td>
<td>550</td>
<td>290</td>
<td>450</td>
<td>93.0</td>
<td>21.5</td>
<td>18.6</td>
<td>9.32</td>
</tr>
<tr>
<td>880</td>
<td>530</td>
<td>350</td>
<td>453</td>
<td>96.0</td>
<td>23.6</td>
<td>19.5</td>
<td>9.77</td>
</tr>
</tbody>
</table>

Table 2: Generator and turbine powers as a function of turbine flowrates.

<table>
<thead>
<tr>
<th>Turbine Volumetric Flowrate (L/s)</th>
<th>Rotational Speed of Turbine (rpm)</th>
<th>Generator Voltage (volts)</th>
<th>Generator Current (amps)</th>
<th>Generator Power (W)</th>
<th>Turbine Power (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.35</td>
<td>73.8</td>
<td>3.20</td>
<td>0.58</td>
<td>1.86</td>
<td>4.81</td>
</tr>
<tr>
<td>4.35</td>
<td>106</td>
<td>4.25</td>
<td>0.70</td>
<td>3.00</td>
<td>6.92</td>
</tr>
<tr>
<td>6.88</td>
<td>166</td>
<td>6.96</td>
<td>0.93</td>
<td>6.49</td>
<td>10.8</td>
</tr>
<tr>
<td>8.14</td>
<td>176</td>
<td>7.38</td>
<td>0.97</td>
<td>7.16</td>
<td>11.4</td>
</tr>
<tr>
<td>9.32</td>
<td>180</td>
<td>7.70</td>
<td>1.00</td>
<td>7.70</td>
<td>11.7</td>
</tr>
<tr>
<td>9.77</td>
<td>192</td>
<td>8.05</td>
<td>1.02</td>
<td>8.21</td>
<td>12.5</td>
</tr>
</tbody>
</table>
CHAPTER 5

ANALYSIS TECHNIQUE

This chapter will elaborate on the analysis used to produce computational results on the performance of the lab-scale WCFT. The two commercial computer codes used to carry out this work are the solid modeling computer aided design software SolidWorks and the computational fluid dynamics software ANSYS Fluent. Other parts of the ANSYS suite of software required are ANSYS Design Modeler and ANSYS Meshing.

The first step in this analysis is to produce a graphic model of the WCFT in SolidWorks. An academic version of SolidWorks 2015 was used to create this three-dimensional model. The SolidWorks model is the means by which geometric information of the WCFT is imported into ANSYS Workbench for the purpose of additional geometry refinement and computational fluid dynamics analysis. Before the analysis of the WCFT model in ANSYS Fluent can be done, the SolidWorks model is imported into ANSYS Design Modeler for required refinement. ANSYS Design Modeler is used to switch from the solid part of the turbine to the region of the turbine where the fluid is located. The fluid domain obtained from ANSYS Design Modeler is meshed using the meshing tool in ANSYS called ANSYS Meshing. Finally, the mesh is imported into ANSYS Fluent where initial and boundary conditions were imposed. A three-dimensional, transient analysis using a finite volume numerical routine and volume of fluid (VOF) model to handle the air and water mixture in the turbine are implemented to perform the simulation of this lab-scale WCFT. Lastly, post-processing of the computational results is done.
5.1. GEOMETRY

To do this analysis, a great deal of geometrical information on the WCFT is required. These were obtained by taking size measurements on the lab-scale WCFT after the experimental data presented in Chapter 4 was acquired. After completion of the experiments, the model WCFT was disassembled to measure the physical sizes of different parts of the turbine like the thickness of the blades, the radius of curvature of the blades, and the orientation of the blades. Similarly, the total length, breadth, and width of the turbine were measured. Standard Allen wrenches were used to disassemble the model so that each of the parts of the turbine could be accessed. A vernier caliper was used to measure the blade thickness and a micrometer screw gauge was used to measure the diameter of the central shaft. A measuring tape and ruler were used to measure the different dimensions of the turbine casing.

With these measurements, a three-dimensional geometry of the WCFT is produced in SolidWorks. The SolidWorks drawing is shown in Figure 16. The turbine blades and casing of the turbine are modeled independently as part drawings and these parts are then combined into a single assembled geometry using the assembly mode in SolidWorks. For the casing, a complete two-dimensional sketch is drawn which is then converted to three dimensions using the extrude feature in SolidWorks. For the blades, a two-dimensional a sketch is produced for a single blade and the central shaft which is then converted to three dimensions using the extrude feature. Finally, twelve or nine blades are produced using the curve pattern feature which duplicates the turbine blades in the desired number along the circumference of the circle diameter of six inches. Then, the final three-dimensional turbine geometry was imported into ANSYS Workbench 17.1, which includes ANSYS Design Modeler, ANSYS Meshing, and ANSYS Fluent for analysis.

This analysis is about fluid flow, but the imported geometry is the solid domain. The region surrounded by the solid is required for a fluid flow analysis. In order to obtain the fluid flow domain, the fill tool in ANSYS Design Modeler is used. The obtained fluid domain is divided into two parts, namely a stationary fluid domain and rotating fluid domain. The rotating fluid domain consists of the volume in and around the periphery of the blades, which possesses rotational motion. The other domain is the stationary fluid domain, which is the remaining flow region after subtracting the rotating fluid domain from total turbine casing. The rotating fluid domain consists of the turbine blades, the rotating walls adjacent to the turbine blades, and the central shaft;
whereas, the stationary fluid domain consists of the inlet, the outlet, and the boundary walls associated with these regions. Finally the total fluid domain was split into two equal halves using the symmetry tool in ANSYS Design Modeler. The symmetry plane is parallel to both walls of the turbine casing and passes exactly halfway between these two walls. The geometry is exactly symmetrical about this plane which reduces the volume of the fluid domain that must be part of the computational domain by a factor of two. This reduces the number of nodes and elements to be solved by the ANSYS fluent solver by a factor of two, and increases the computational speed by a factor of two. This final geometry from ANSYS Design Modeler is then imported into the ANSYS Meshing routine.

Figure 16: SolidWorks drawing of WCFT.
5.2. MESHING

Meshing is an important step in any computational fluid dynamic simulation. The tool used for meshing the WCFT geometry is ANSYS Meshing. The geometry of the turbine is complex with a number of curved blades; however, the two opposite faces of the turbine casing share the same topology which makes it possible to use the sweep feature in ANSYS Meshing. In this technique, one of the two faces sharing the same topology, called the source face, is meshed. The sweep feature in ANSYS Meshing then advances this mesh across the turbine casing volume to the other wall. Since the fluid domain is symmetrical along the z-axis, only half of the total fluid domain is meshed in order to save computational time. Figure 17 shows the symmetrical mesh structure where the positive x-axis is horizontal and points to the right, positive y-axis is vertical and points upward, and the positive z-axis comes out of the page. The stationary and rotating fluid zones were meshed separately, but both of them use the sweep feature. The element size of the mesh is controlled by a global size function which takes the user input for minimum element size and maximum face size. These sizes are used as basic parameters in determining the number of cells used. In order to increase or decrease the number of cells, the size of element is increased or decreased. The inlet, outlet, blades, walls, stationary fluid domain, and rotating fluid domain are assigned different names in order to make it easier to apply initial conditions and boundary conditions in the subsequent step.

5.2.1. Mesh Method

The sweep method is one of the most efficient meshing techniques available in ANSYS Meshing. This method of meshing is designed in such a way that it introduces mostly hexahedral elements along with some wedge-shaped elements in the fluid domain. Since hexahedral elements result in more structured grids, they provide more accurate results as compared to elements that tend to provide unstructured grids. But the fact is, the sweep method is geometry dependent and cannot be applied to every geometry. As previously explained, the turbine geometry is a sweepable body and both fluid domains are meshed separately using this single method. The formation of hexahedral elements makes the total count of elements less, the time used in creating the mesh is smaller, and the computational time is smaller. The selection of a source and target face was done manually in order to create inflation near the walls; this is impossible with the automatic
source/target selection. More than ninety percent of the total elements formed were hexahedral, while the remaining were wedged.

Figure 17: WCFT mesh.

5.2.2 Near Wall Meshing

The presence of walls significantly affects the turbulence of the flow and near-wall modeling considerably affects the reliability of the solution. The near-wall region’s variable gradients are high and the transport of quantities like momentum occur robustly. In order to capture these near wall effects with the numerical solution routine, layers of inflation are developed near all wall regions. Inflation meshing is using very small size cells next to wall and in boundary layers which increase in size as the distance from the wall increases.

In order to calculate the optimum thickness of the inflation layer, a suitable first layer grid size is needed. This grid size is set equal to the distance from the wall where the non-dimensional wall coordinate
\[ y^+ = \frac{ypU_\tau}{\mu_k} \]  
(5.1)
is equal to 1.0. Equation (5.1) is solved for the dimensional distance from the wall, \( y \), where \( y^+ = 1.0 \). In order to calculate the first layer grid size, the density of the fluid, \( \rho \), the shear velocity, \( U_\tau \), and the dynamic viscosity, \( \mu_k \), are required. In this work, liquid water at approximately room temperature is considered and thus \( \rho = 998 \ \text{kg/m}^3 \) and \( \mu_k = 8.90 \times 10^{-4} \ \text{kg/m-s} \). To determine the frictional velocity the equation

\[ U_t = \sqrt{\frac{\tau_w}{\rho}}, \]  
(5.2)
is used where the shear stress at the wall, \( \tau_w \), is obtained from

\[ \tau_w = \frac{1}{2} C_f \rho U_\infty^2. \]  
(5.3)
The friction coefficient, \( C_f \), is determined from the following formula \[36]\)

\[ C_f = 0.058 \frac{Re}{Re^{-0.2}} \]  
(5.4)

where the Reynolds number is defined as

\[ Re = \frac{\rho UD_h}{\mu_k}. \]  
(5.5)
The equation for the hydraulic diameter is

\[ D_h = \frac{4A}{P}, \]  
(5.6)
The hydraulic diameter is calculated assuming the flow channel between the turbine blades is a rectangular duct and thus the hydraulic diameter becomes

\[ D_h = \frac{2ab}{a+b}, \]  
(5.7)
where \( a \) and \( b \) are the height and width of the rectangular duct.

Doing these calculations for a typical WCFT operating condition results in a wall grid layer height of 0.01 mm. As seen in Figure 18, ten layers of inflation were produced around all the blade walls; and as seen in Figure 19, twenty layers of inflation were produced around the rest of the walls. A default growth rate of 1.2 and transition ratio of 0.272 are used for all inflations. With the
Figure 18: Inflation cell layers near blade walls.

Figure 19: Inflation cell layers near stationary wall.
introduction of inflation, the number of cells increase, while the quality of the cells decreases. For a turbine with nine blades, the addition of inflation near the walls increases the number of elements from 732640 to 926166. Further, the addition of inflation decreased the minimum orthogonal quality from 0.72 to 0.27 and increased the maximum skewness ratio from 0.56 to 0.84. Although inflation increased the computational time and decreased the quality of the mesh, the increase in accuracy obtained is well worth the cost of inflation.

5.3. GOVERNING DIFFERENTIAL EQUATIONS

5.3.1. Governing Equations Using the Volume of Fluid (VOF) Technique

This work uses an Euler-Euler mixture model to simulate different fluids in the same computational domain. In the Euler-Euler approach, the different fluids are treated as interpenetrating, continuous fluids. The volume fractions are taken to be a continuous function of space and time and the volume fractions of all fluids in each control volume sum to unity. All control volumes must be filled with liquid water or air and void regions are not allowed. With the VOF model, one vector momentum equation is solved using properties based on the fluid mixture present in each cell. The volume fraction of both fluids, in each cell, are tracked and calculated with a conservation of mass equation.

Under most operating conditions the WCFT will have both air and water within its casing. The location of these two fluids within the casing is not known ahead of time; and thus, both fluids must be tracked by the analysis. This work, simultaneously tracks two fluids, water and air within the fixed computational volume of the casing of the WCFT (see Figure 17). These two fluids are immiscible and have a clearly defined interface which makes them suitable for this model. A general continuity equation used in finding the volume fraction of water is

$$\frac{1}{\rho_w} \frac{\partial}{\partial t} (\alpha_w \rho_w) + \frac{1}{\rho_w} \nabla \cdot (\alpha_w \rho_w \vec{v}_w) = \frac{S_{aw}}{\rho_w} + \frac{1}{\rho_w} \sum_{p=1}^{n} (\dot{m}_{aw} - \dot{m}_{wa}),$$ (5.8)

where $\alpha_w$ is the fraction of volume occupied by water and this quantity can fluctuate between zero and one. The volume fraction of air is not calculated using a continuity equation; but rather is calculated using the constraint that the sum of the volume fractions of water and air is unity. In Equation (5.8), $S_{aw}$ is a water source term that accounts for other species changing into water and $\dot{m}_{aw}$ and $\dot{m}_{wa}$ are meant to handle phase change phenomena where $n$ is the number of these phase
change phenomena occurring. Since the WCFT problem being considered here, does not have any chemical reactions occurring where one species is converted to another, the $S_{aq}$ term on the right side of Equation (5.8) is clearly zero. Moreover, there is no phase change phenomena between water and air which nullifies $\dot{m}_{aw}$ and $\dot{m}_{wa}$ terms. There is some water that flashes into water vapor within the casing of the WCFT, but this is so small and insignificant that there is no reason to include this in the analysis. The version of Equation (5.8) solved for this work is

$$\frac{1}{\rho_w} \frac{\partial}{\partial t} (\alpha_w \rho_w) + \frac{1}{\rho_w} \nabla \cdot (\alpha_w \rho_w \vec{v}_w) = 0.$$  \hspace{1cm} (5.9)

From this equation, volume fractions of water are computed in each control volume and the volume fractions of air are simply

$$\alpha_a = 1 - \alpha_w. \hspace{1cm} (5.10)$$

The momentum equation,

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \left[ \mu (\nabla \vec{v} + \nabla \vec{v}^T) \right] + \rho \vec{g} + \vec{F},$$ \hspace{1cm} (5.11)

and the conservation of mass equation,

$$\frac{\partial}{\partial t} (\rho) + \nabla \cdot (\rho \vec{v}) = 0,$$ \hspace{1cm} (5.12)

are influenced by the volume fractions of air and water through the properties $\rho$ and $\mu$ as

$$\rho = \alpha_w \rho_w + \alpha_a \rho_a$$ \hspace{1cm} (5.13) and

$$\mu = \alpha_w \mu_w + \alpha_a \mu_a.$$ \hspace{1cm} (5.14)

Turbulent shear stresses are included in Equation (5.11) using the Boussinesq approximation and the shear stress transport (SST) $k - \omega$ turbulence model. The SST $k - \omega$ turbulence model is discussed below and the Boussinesq approximation is

$$turbulent \ shear \ stresses = \nabla \cdot \left[ \mu_t (\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \vec{k} \vec{l} \right]$$ \hspace{1cm} (5.15)

Equation (5.15) can be included in Equation (5.11) through the $\vec{F}$ term. The quantity $\nabla \vec{v}^T$ in Equations (5.11) and (5.15) is a dyadic product of the gradient vector times the transpose of the velocity vector which results in a second order tensor. The term $[\mu (\nabla \vec{v} + \nabla \vec{v}^T)]$ as a whole models all the molecular viscous forces present in the flow, except those for very strong compressibility
effects. These are not important for the low speed flows considered here. The quantities $p$ and $\mathbf{g}$ in Equation (5.11) are the pressure and gravitational acceleration, respectively. The quantity $S_m$ in Equation (5.12) is a mass source term and is taken as zero in this work. In this work there is a strong coupling between the momentum equation, the overall conservation of mass equation, the water species conservation of mass equation, and the turbulence equations to be discussed below.

5.3.2. Turbulence Model

As mentioned above the turbulence model used in this work is the SST $k - \omega$ turbulence model. This model was developed by Menter in August 1994\cite{Menter} and is one of the most widely used turbulence models. The SST $k - \omega$ model differs from the standard $k - \omega$ model in a few ways. Probably the biggest difference is the SST version of the $k - \omega$ model is actually a blend of an updated $k - \omega$ model and a transformed $k - \varepsilon$ model. This blending is done such that the $k - \omega$ model is used in the inner portion of the boundary layer and the $k - \varepsilon$ model is used in the outer portion of the boundary layer. A natural and smooth transition is made between these two models. Other differences between the SST $k - \omega$ model and the standard $k - \omega$ model are that the SST version uses a different turbulent viscosity formulation, it uses different constants, and it adds an additional term to the dissipation equation. These updates to the standard $k - \omega$ model make the SST $k - \omega$ model more accurate and applicable to a wider range of flow situations. The equation set shown below on the SST $k - \omega$ model are taken from the Fluent manual\cite{FluentManual}.

5.3.2.1. Governing Differential Equations

The SST $k - \omega$ model is a two equation model that accounts for the turbulent kinetic energy, $k$, and dissipation rate of this energy, $\omega$. The differential equation for the turbulent kinetic energy is

\[
\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho k \mathbf{v}) = \nabla \cdot (\tau_k \nabla k) + G_k - Y_k + S_k, \tag{5.16}
\]

and that for the dissipation rate of this energy is

\[
\frac{\partial}{\partial t} (\rho \omega) + \nabla \cdot (\rho \omega \mathbf{v}) = \nabla \cdot (\tau_\omega \nabla \omega) + G_\omega - Y_\omega + D_\omega + S_\omega. \tag{5.17}
\]
In both of these equations, the first terms on the left-hand side represent the rate of change of $k$ or $\omega$ and second terms represent convective transport of $k$ or $\omega$. The first terms on the right-hand side of these equations represent diffusion of $k$ or $\omega$ where $\tau_k$ and $\tau_\omega$ represent the effective diffusivity of each of these quantities. The quantities $G_k$ and $G_\omega$ represent production of turbulent kinetic energy due to mean velocity gradients and the production of the specific dissipation rate respectively. $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$ because of turbulence and $S_k$ and $S_\omega$ are the user defined source terms; neither of which is used in this work. The additional term $D_\omega$, in the equation for the dissipation of turbulent kinetic energy, is a cross diffusion term that was not present in the standard $k - \omega$ model.

5.3.2.2. Effective Diffusivity Terms

The effective diffusivity terms $\tau_k$ and $\tau_\omega$ are calculated with

$$\tau_k = \mu + \frac{\mu_t}{s_k}$$ (5.18)

and

$$\tau_\omega = \mu + \frac{\mu_t}{s_\omega}$$ (5.19)

In the above formulas, $s_k$ and $s_\omega$ are the turbulent Prandtl numbers for $k$ and $\omega$ given as

$$s_k = \frac{1}{F_1/s_{k,1} + (1 - F_1)/s_{k,2}}$$ (5.20)

and

$$s_\omega = \frac{1}{F_1/s_{\omega,1} + (1 - F_1)/s_{\omega,2}}$$ (5.21)

and $\mu_t$ is the turbulent viscosity given by

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max[\frac{1}{2}SF_2, 0]}.$$ (5.22)

In the above equations, $S$ is the strain rate magnitude

$$S = \sqrt{(\nabla \vec{v} + \nabla \vec{v}^T) \cdot (\nabla \vec{v} + \nabla \vec{v}^T)}$$ (5.23)

and $F_1$ and $F_2$ are the blending functions given as

$$F_1 = \tanh(\frac{\varphi_1^4}{})$$ (5.24)
and \[ F_2 = \tanh(\varphi_2^2). \] (5.25)

The required quantities in the blending functions are
\[
\varphi_1 = \min \left[ \max \left( \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right), \frac{4\rho k}{\sigma_{\omega,2}D_{\omega}^+ y^2} \right] \] (5.26)

where
\[
D_{\omega}^+ = \max \left[ 2\rho \frac{1}{\sigma_{\omega,2}} \frac{\omega}{\omega \delta_{x_j} \delta_{x_j}}, 10^{-10} \right] \] (5.27)

and
\[
\varphi_2 = \max \left[ 2 \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2} \right]. \] (5.28)

The \( \alpha^* \) term in Equation (5.22) is determined with
\[
\alpha^* = \alpha_{\infty}^* \left( \frac{\alpha_0^* + \frac{Re_t}{R_k}}{1 + \frac{Re_t}{R_k}} \right) \] (5.29)

where
\[
\alpha_0^* = \frac{\beta_i}{3}, \] (5.30)
\[
\alpha_{\infty}^* = F_1 \alpha_{\infty,1} + (1 - F_1)\alpha_{\infty,2}, \] (5.31)
\[
\alpha_{\infty,1} = \frac{\beta_{l1}}{\beta_{\infty}} - \frac{k^2}{\sigma_{\omega,1} \beta_{\infty}^*}, \] (5.32)
\[
\alpha_{\infty,2} = \frac{\beta_{l2}}{\beta_{\infty}} - \frac{k^2}{\sigma_{\omega,2} \beta_{\infty}^*}, \] (5.33)

and
\[
Re_t = \frac{\rho k}{\mu \omega}. \] (5.34)

For Equations (5.20-5.22) and Equations (5.26-5.33) a number of constants are required. For the SST \( k - \omega \) model these constants are \( \sigma_{k,1} = 1.176, \sigma_{\omega,1} = 2.0, \sigma_{k,2} = 1.0, \sigma_{\omega,2} = 1.168, \alpha_1 = 0.31, R_k = 6, \alpha_{\infty}^* = 1.0, \beta_i = 0.072, \beta_{l,1} = 0.075, \beta_{l,2} = 0.0828, \kappa = 0.041 \) and \( \beta_{\infty}^* = 0.09 \). These constants allow the solution of the effective diffusivity terms \( \tau_k \) and \( \tau_\omega \) and the turbulent viscosity, \( \mu_t \).
5.3.2.3. Production Terms

$G_k$ and $G_\omega$ are the terms which represent the production of turbulent kinetic energy and production of the specific dissipation rate in Equations (5.16) and (5.17). The turbulent kinetic energy production is determined from

$$G_k = -\rho \overline{u_i' u_j'} \frac{\partial \overline{u_j}}{\partial x_i}$$

(5.35)

and the specific dissipation rate production from

$$G_\omega = \frac{\alpha}{\nu_t} G_k.$$  

(5.36)

Here, $G_k$ comes from Equation (5.35) and the coefficient $\alpha$ is represented by

$$\alpha = \frac{\alpha_\infty}{\alpha^*} \left( \frac{\alpha_0}{1 + \frac{Re_t}{Re_\omega}} \right)$$

(5.37)

where $Re_\omega = 2.95 \alpha^*$, $\alpha_0 = \frac{1}{9}$, and $\alpha^*$ damps the turbulent viscosity and is given by Equation (5.29) where $Re_t$ is given by equation 5.34.

5.3.2.4. Dissipation Terms

$Y_k$ and $Y_\omega$ are the dissipation terms in the original equation of the SST k-$\omega$ turbulence model. Dissipation of $k$ is given by

$$Y_k = \rho \beta^* k \omega$$

(5.38)

where

$$\beta^* = \beta_1^* [1 + 2 F(M_t)],$$

(5.39)

$$\beta_1^* = \beta_\infty \left( \frac{4}{15} \left( \frac{Re_t}{R_\beta} \right)^4 \right),$$

(5.40)

and

$$Re_t = \frac{\rho k}{\mu \omega}.$$  

(5.41)

The constants required to obtain the dissipation terms are $2^* = 1.5$ and $R_\beta = 8$. Because the flows in this work are relatively slow, the compressibility function $F(M_t) = 0$. 

48
Similarly, dissipation of $\omega$ is represented by $Y_\omega$ in the main turbulent equation and the formula is

$$Y_\omega = \rho \beta \omega^2$$  \hspace{1cm} (5.42)

where

$$\beta = \beta_i \left[ 1 - \frac{\beta_i^*}{\beta_i} \frac{1}{F(M_e)} \right]$$  \hspace{1cm} (5.43)

and

$$\beta_i = F_1 \beta_{i,1} + (1 - F_1) \beta_{i,2}.$$  \hspace{1cm} (5.44)

The term $F_1$ is obtained from Equation (5.24). The cross-diffusion term is represented by $D_\omega$ in Equation (5.17) and is given by

$$D_\omega = 2(1 - F_1) \rho \frac{1}{\omega \sigma_{w,2}} \nabla k \cdot \nabla \omega.$$  \hspace{1cm} (5.45)

5.4. NUMERICAL TECHNIQUE

5.4.1. Velocity and Pressure Field Solver

Two solver options are available in ANSYS Fluent, namely the pressure based solver and the density based solver. The density based solver was specifically designed for high velocity, compressible flows. Due to the incompressible flow character of the WCFT situation, the pressure based solver is chosen for this analysis. In this method, the finite-volume discretization process is employed where the governing equations are solved for each discrete computational control volume. In this technique the vector momentum equation is written as a three momentum equations, one for each coordinate direction, which are arranged to solve for each of the three velocity components. The mass conservation equation is arranged to solve for the pressure field. These equations are coupled to each other and to the other governing differential equations required for this analysis; and a trial and error solution is required to solve them. Figure 20 shows the general flowchart of this solution process.

Two options for pressure discretization are available in Fluent for use with the VOF method, namely PRESTO and body-force-weighted schemes. PRESTO, otherwise known as pressure staggering option, is opted for in this work because it is suitable for flows with high swirl numbers. In this scheme, the control volume face pressure is calculated by using a staggered grid
set of discrete control volumes. Having the pressure grid staggered from the velocity grids eliminates a number of numerical difficulties.

Figure 20: Flowchart of pressure based solution method.[38]

5.4.3 Volume Fraction Solver

Implicit and explicit volume fraction formulations are available in ANSYS Fluent. The explicit formulation is non-iterative in the time direction, whereas the implicit formulation is iterative. Since the numerical accuracy of an explicit formulation is better than the implicit formulation and it is noniterative, the former was chosen at the start of this research. However, it was soon realized that time steps larger than those allowed by the explicit formulation were
beneficial. For this reason, a change was made to the implicit formulation, which is unconditionally stable. All the results presented as part of this thesis work were done using an implicit volume fraction formulation.

5.4.3.1. Interface Modeling

The number of Eulerian phases present in the WCFT are two: air and water. The compressive scheme of the volume fraction method is selected for interface modeling. This is a second order reconstruction scheme based on a slope limiter. For the compressive scheme, the value of the slope limiter is two.

![Figure 21: Actual interface (left) and interface from compressive formulation (right).](image)

Figure 16 above portrays the comparison between the actual interface and the interface obtained with this scheme. As shown in this figure, this is a sharp interface modeling scheme which is capable of obtaining a clear and distinct interface between the phases. The volume fraction cut-off point is selected as $1 \times 10^{-6}$. This means any water volume fraction values below this limit are automatically set to zero. Similarly, any water volume fraction values higher than $1.0 - 1 \times 10^{-6}$ are automatically set to one.
5.4.5. Turbulence Solver

A second order upwind discretization scheme is used for the turbulent kinetic energy and turbulent dissipation rate equations. ANSYS Fluent assigns discrete values of every scalar quantity at the center of the control volume; however, face values are required for convection terms and these face values are interpolated using cell center values. In the second order upwind scheme, cell face quantities are computed using a multidimensional linear reconstruction approach. A Taylor series expansion of the cell centered solutions to the cell faces is used to achieve higher order accuracy in the solution.
CHAPTER 6

COMPUTATIONAL RESULTS

In this chapter results for the performance of the lab-scale WCFT are presented. A number of field quantities are presented that help the reader to understand the performance of these turbines, but the primary results are power versus rotational speed. Since the reason for installing a WCFT at the base of a low head dam is to produce power, shaft power output from these computational simulations is the result that is desired the most. Turbine shaft power varies as a function of the characteristics of the load put on a turbine. This external load can be an electric generator or even the lifting of a weight. The interest of this study is to utilize WCFTs for generating electricity; thus, the load used in the experimental portion of this work is an electric generator attached to some lights. In the computational portion of this work it can be imagined that the turbine is connected to an electric generator, but no electric generators are simulated in this work. The way in which a load external to the WCFT is included in the simulated results, is to run the simulation for a given water flowrate at many turbine rotational speeds. Higher rotational speeds represent lower torque loads on the turbine shaft and lower rotational speeds represent higher torque loads on the rotational shaft. Tying this back to the experimental work that was done, higher turbine rotational speeds correspond to less lights in Figure 10 and lower rotational speeds correspond to more lights. The difficulty in obtaining comprehensive turbine performance for one turbine design is, computational runs at several different turbine rotational speeds must be produced. This is difficult because one rotational speed takes several days of computational time. For a computer with RAM of 4 GB and Intel Core i5 CPU of 3.20 GHz, average computational time required to obtain the output is more than four days. More than 250 computational runs were
completed before acquiring final results which would have taken more than 2 years of time if done using a conventional computer. Hence, in order to save time, the Ohio Super Computer (OSC) was chosen for computational analysis. With the aid of a supercomputer, computational time for a single simulation was reduced to approximately 20 hours. The number of processors used were 12, which allowed parallel computing thereby decreasing the time significantly.

The field results obtained as part of this work are complete and cover the entire fluid region in the turbine; however, results are only presented at the center plane of the turbine. The field results presented are volume fraction, different types of velocity plots, density, and pressure. These plots help to inform the reader of what is occurring inside the turbine. The field plots presented in this thesis provide results that would be extremely difficult, if not impossible, to obtain experimentally. Hopefully these field plots provide insights into better turbine design.

So that the reader can have confidence in the computational results presented in this thesis, two things are done. First a grid and time step independence study is performed; and second one of the experimental cases shown in Table 1 is simulated and the results compared. It is hoped that these two activities provide the reader with confidence in the computational results presented here.

The grid study, the time step study, and the validation study are done under what will be called base case operating conditions. The base case operating conditions are shown in Table 3. These are the same operating conditions as the third case shown in Tables 1 and 2. This case is in the middle of the water flowrates tested experimentally.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Numerical Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>Head from center of turbine</td>
<td>15.8</td>
<td>cm</td>
</tr>
<tr>
<td>Turbine water flow rate</td>
<td>6.88</td>
<td>L/s</td>
</tr>
<tr>
<td>Rotational speed of turbine</td>
<td>166</td>
<td>rpm</td>
</tr>
<tr>
<td>Turbine output</td>
<td>10.8</td>
<td>W</td>
</tr>
<tr>
<td>Generator output</td>
<td>6.48</td>
<td>W</td>
</tr>
</tbody>
</table>

*Table 3: Base case conditions from experimental measurements.*
6.1. GRID AND TIME STEP INDEPENDENCE STUDIES

6.1.1. Grid Independence Study

To be sure that the final computed results are not dependent on the mesh used to perform an analysis, a grid independence study is done. In order to do this study, the total fluid domain was divided into different numbers of elements (also called control volumes and cells) and simulations run for each mesh size. Different mesh sizes are produced using ANSYS meshing. In this study, the maximum number of nodes and elements generated are 2,356,575 and 2,246,144, respectively, resulting in the smallest element sizes; and the minimum numbers are 131,426 and 139,197, respectively, resulting in the largest element sizes. It must be realized that many different element sizes are used throughout the computational domain for any given simulation. Thus, in the 131,426 element run, there are some small elements and some larger elements. Mesh inflation is used in all the grid independence simulations. As the element size is decreased and the number of nodes and elements are increased, grids are refined equally in every direction. The number of nodes and elements and the maximum element size obtained for all the runs done as part of this grid independence study are presented in tabular form in Table 4. The time step size used for all the grid convergence study data points is 0.0005 seconds.

Given in Table 4 in numerical form and in Figure 21 graphically are the results of this grid independence study. The output parameter used to determine grid convergence is the average power produced by the turbine. The graphical representation of these results clearly shows a convergence trend as the number of grid points are increased. This plot indicates that reasonable convergence is obtained above 750,000 grid points. A more quantitative understanding of the convergence results can be obtained by looking at the results in the last column of Table 4. Taking the last average turbine power in this table as the precise power, it can be determined that 652,756 grid points provides convergence within 6.0%, 876,049 grid points provides convergence within 3.2%, and 1,351,036 grid points provides convergence within 1.8%. Thus, results will be taken as being converged within engineering accuracy when 1,351,036 grid points are used. Given the many difficulties in doing these types of calculations, any grid using more than 1 million points is acceptable. In this work, the number of grid points used for the rest of the results presented in this thesis is in the neighborhood 1.3 million. Different blade geometries will result in slightly different grid numbers.
Table 4: Power as a function of the mesh used.

<table>
<thead>
<tr>
<th>Maximum Element Size (mm)</th>
<th>Number of Nodes</th>
<th>Number of Elements</th>
<th>Power (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.35</td>
<td>131,426</td>
<td>139,197</td>
<td>0.956</td>
</tr>
<tr>
<td>3.00</td>
<td>245,646</td>
<td>264,891</td>
<td>3.680</td>
</tr>
<tr>
<td>2.50</td>
<td>416,818</td>
<td>441,914</td>
<td>6.873</td>
</tr>
<tr>
<td>2.00</td>
<td>652,756</td>
<td>606,943</td>
<td>8.474</td>
</tr>
<tr>
<td>1.75</td>
<td>933,918</td>
<td>876,049</td>
<td>8.718</td>
</tr>
<tr>
<td>1.50</td>
<td>1,429,224</td>
<td>1,351,036</td>
<td>8.851</td>
</tr>
<tr>
<td>1.25</td>
<td>2,356,575</td>
<td>2,246,144</td>
<td>9.013</td>
</tr>
</tbody>
</table>

Figure 22: Grid independence study results.
6.1.2. Time Step Independence Study

Because of the rotation of the WCFT blades and the unsteady manner in which the two phase fluid moves through the WCFT rotor, these calculations are performed including unsteady terms in all the governing differential equations shown in Chapter 5. To demonstrate that the final time averaged results are independent of the time step used, a time step independence study is performed. As with the grid independence study, the output parameter used to prove convergence is the time averaged output power of the turbine. Every input parameter used throughout the time step study is that for the base case shown in Table 3, except the time-step size. The mesh used for the entire time step study is 1,351,036 million elements. Six time-steps were considered starting from 0.0002 seconds up to 0.0032 seconds with a twofold increase between each step.

Shown in Figure 23 is the average output power plotted against time-step size. In general, as the time step size decreases the calculated output power tends to converge, with an unexpected variation seen for the smallest time step of 0.0002 seconds. The 0.0002 second result drops 1.2% from the 0.0004 second result. It was expected that the 0.0002 second result would level off with the 0.0004 second result. As Figure 23 shows, it actually dropped. It is felt that truncation error is responsible for the drop seen between the 0.0002 second result and the 0.0004 second result. For this reason time steps less than 0.0004 seconds are not used. A time step of 0.0005 seconds is chosen to use for the rest of the computational simulations done as part of this thesis work. Because the 0.0004 second result is 3.6% higher than the 0.0008 second result, it is estimated that a time step of 0.0005 seconds converges results within 4%. The possibility for better time step convergence does not exist, unless a higher numerical precision is utilized. Because of the computational costs associated with going to higher numerical precision, this was not done.

6.2. COMPUTER MODEL VALIDATION

The experimental case chosen to validate the computational results is the third case shown in Table 1 and shown again in a different manner in Table 3. This is the base case and the base case uses 12 blades for the rotor of the turbine. To perform this comparison, a key input to the computational analysis is the velocity of the water. This velocity comes from two aspects of the flow over the dam. The first aspect is the horizontal velocity the water has as it flows over the dam. This velocity is obtained from
where $A_h$ is the cross sectional area of the water flowing over the dam (see Figure 15). This was measured as accurately as possible with the results being $2.5 \text{ cm} \times 30 \text{ cm} = 75 \text{ cm}^2$. The width of the dam is relatively easy to determine, it is the same as the width of the water channel; however, the height is a little more difficult to determine because it changes as the water flows over the dam. For a volumetric flow rate of 13.8 L/s, Equation (6.1) provides a horizontal velocity component of 1.84 m/s. The second aspect of the turbine inlet velocity is the acceleration of the water as it drops from the top of the turbine to the entrance of the turbine. This acceleration results in a vertical velocity component of the water entering the turbine as

$$v_v = \sqrt{2gh}.$$  \hspace{1cm} (6.2)

Using a drop height of 15.8 cm from the top of the dam to the inlet of the turbine produces a vertical velocity of 1.76 m/s. These two velocity components serve as the input boundary condition of the water flow into the WCFT.
The validation simulation was carried out under these circumstances and ANSYS Fluent determined torques on the turbine rotor as a function of time. This torque data was exported to an EXCEL spreadsheet where it was converted to powers using

\[ P = T \omega \]  

(6.3)

where \( T \) is torque and \( \omega \) is the angular rotational speed of the rotor. The angular rotational speed of the rotor can be obtained from the cyclical rotational speed in rpm with

\[ \omega = \frac{2\pi (\text{rpm})}{60}. \]  

(6.3)

For this operating case the rotational speed of the rotor is 166.38 rpm.

As can be seen from Figure 24 the instantaneous power results oscillate rapidly. This is due to the rotation of the blades and how the blades and channels between the blades line up with the incoming water flow. The relative arrangement of the blades to the incoming water changes 12 times per revolution of the blades. On average the instantaneous power results increase from about zero, reach a maximum value, and decay to steady state behavior. It the average of this steady state behavior that is used as the computed power of the WCFT. The y-axis location of the brown straight line is this average and the horizontal length of this straight line is the time frame used to obtain this average. The calculated average output power is 8.85 watts which should be compared to the 10.8 watt value obtained experimentally. This means the calculated power is 18.1% lower than the experimentally determined power. Although 8.85 watts is hydraulic power whereas 10.8 watts is brake power, this is the best comparison approximation that can be done from computational analysis. Hence, the hydraulic power is compared with the brake horse power. This is considered a reasonable comparison given the many complications that exist in this analysis.

To further validate the computation a mass imbalance check was performed. Since there is no mass generation inside the WCFT, every discrete control volume within the computational domain should have the mass flowing in equal to the mass flowing out. The difference between the mass flowing in and the mass flowing out will be called the mass imbalance, and a large mass imbalance represents a problem in the computation. Figure 25 shows the mass imbalance histogram for a simulation of the WCFT. This figure indicates that conservation of mass is being maintained across the computational domain within \( 5 \times 10^{-8} \) kg/s.
6.3. FIELD RESULTS

In this section detailed results of a number of quantities across the mid-plane, symmetry plane at \( z = 3 \) inches, of the WCFT are presented. These results provide insight to the flow field within the turbine. All the results in this section are for a 9-blade turbine where the rotation of the turbine rotor is clockwise at 150 rpm. The volume flowrate for this case is 6.88 L/s. These results are taken at one instant of time during the steady state operation of the turbine. This means these results are for one orientation of the blades relative to the incoming water flow. This one orientation occurs 9 times during one rotation, but for different blade incoming water flow orientations, different results will be obtained. The oscillations in Figure 24 show this.

Figure 24: Instantaneous output power for base case situation.
6.3.1. Volume Fractions

Figure 26 provides the reader with information on where the water is located in the turbine and where the air is located in the turbine. Since this figure gives the volume fraction of water, one minus this volume fraction is the volume fraction of air. In Figure 26, a red color indicates a volume fraction of one for water and zero for air; while a blue color indicates a volume fraction of zero for water and a volume fraction of one for air. This figure shows how the air and water are mixing and that the air is the dominate fluid in the top half of the turbine and water is the dominate fluid in the bottom half of the turbine. Mixing between the water and the air takes place in the blade channels on the bottom of the rotor and along the outer periphery of the blades at the top of the rotor. The mixing regions of air and water are inherently unsteady and will change from instant to instant. While the results in Figure 25 will change somewhat from one instant to the next, this snapshot in time gives a good representation of where the air and water are located in the turbine. This figure also clearly shows the region where the water from the dam is entering the turbine casing along
the slanted wall of the casing on the right-hand side of the plot. It can be seen that this water moves into two of the blade channels and goes across the turbine exiting out blade channels on the other side of the turbine rotor. Velocity figures given latter will further strengthen this conclusion.

![Image of water volume fraction contours]

Figure 26: Contours of water volume fraction. The air volume fraction is one minus the water volume fraction.

6.3.2. Velocities

With the velocity vector plots shown in this section the reader will be able to ascertain the magnitude and direction of the flow, for both air and water. Five velocity vector plots are given in this subsection, and each of these is colored differently to convey different information. All five of these plots have the overall velocity vectors where the length of the vector indicates the overall velocity magnitude. The longer the vector the higher the speed of the flow. The color of the vector is meant to convey additional information above that conveyed by the velocity vectors.
The first velocity vector diagram is shown in Figure 27. This velocity vector diagram is colored by the magnitude of the velocity. It is recognized that this is the same information conveyed by the length of the vectors, but coloring provides a more quantitative view of the velocity magnitude; while vector lengths provide more of a qualitative view of the velocity magnitude. Figure 27 clearly shows that the highest velocities in the flow are in the upper left region of the casing where the velocity vectors are red in color. This means these velocities are close to 5 m/s. Another interesting flow velocity region is at the water inlet. These velocities are in the 2.0 m/s neighborhood. Most of the fluid in the turbine has a velocity magnitude below 1 m/s.

Inflation near boundary walls and blades resulted in no slip boundary condition near all the walls and development of a velocity profile near the walls is clearly seen when Figure 27 is enlarged in the near wall regions. Due to acceleration from gravity, the velocity of flow was found to increase gradually from inlet of turbine towards the rotating blades and then decreased after striking with blades. The inlet velocity condition was 1.89 m/s but the maximum velocity in the flow domain was found to be 15.49 m/s. This velocity is not the velocity of water flow, but the velocity of air in the upper portion of the outlet region.

The most important thing that can be elucidated from Figure 27 is the overall flow patterns. A lot of the space in the turbine casing is occupied by low speed flow, flows below 0.5 m/s with a very high velocity flow in the upper left corner of the turbine casing. The green areas in this plot show the critical flow patterns in the turbine. The green arrows come down from the inlet along the slanted wall and cut across the rotor below the axis of the rotor. Some of the green arrows indicate flow going out of the blades before reaching the other side of the rotor; and some of the green arrows indicate flow being caught in the turbine blades and traveling to the top of the turbine rotor. The next velocity diagram will show that the green arrows in Figure 27 are indicating water flow. This can also be determined by looking at Figure 26.

So that the reader has a better idea of which velocity vectors apply to water and which apply to air, Figure 28 has been prepared. Figure 28 displays the velocity vectors colored by density, where blue represents the density of air and red represents the density of water. In between colors represent regions where both air and water coexist, and an average density is given. As seen clearly in this figure, the primary path of the water through the WCFT can be seen. The nice cross
flow of the water just below the center of the rotor is clearly seen. This means the turbine is truly acting like a cross flow turbine. It is also seen that some water turns downward and flows out the bottom of the rotor before hitting the rotor blades at a second location, and some water gets caught in the blades and is carried to the top of the rotor. Water that does not follow the design flow path, that is across the turbine rotor, probably lowers the efficiency of the turbine. At the exit of the turbine casing, it can be seen that the air is on top and the water is on the bottom. The air is moving at a higher velocity than the water.

![Velocity vectors colored by velocity magnitude](image.png)

Figure 27: Velocity vectors colored by velocity magnitude where the velocity magnitudes are given in m/s.

Figure 29 is a histogram which depicts the sample velocity distribution of fluid within the turbine. Although the maximum velocity in the fluid domain is a little higher than 15 m/s, more than ninety five percent of the velocities by volume lie below 3 m/s and more than ninety eight percent lie below 6 m/s. This histogram includes both the velocity of air and water. As seen in the
Figure 28: Velocity vectors colored by density where the densities are given in kg/m³.

Figure 29: Velocity histogram.
velocity vectors shown in Figure 28, the higher velocities correspond to air; whereas, the lower velocities correspond to water. The inlet velocity for this case is 1.89 m/s so it makes sense that the majority of velocities present in the analysis are under 3 m/s.

Figures 30 and 31 are the overall velocity vectors colored with velocity magnitudes in the x and y directions, respectively. These figures are presented so the reader can see the magnitudes of the velocity components. The z-direction velocity components are not shown because these results are for a symmetry plane that cuts the turbine in two halves along the z-axis. Just like all the results presented in this thesis, the positive x-axis direction is directed to the right side of the page, the positive y-direction axis is in the same plane as the x-axis and points upward, and the positive z-axis is perpendicular to x and y axis coming out of paper. Since the actual flow in the turbine is three-dimensional, the fluid is free to flow in any direction and fluid velocities, in general, have three components. The computation is also three dimensional and captures the three dimensional nature of this flow. The maximum velocities in the x and y directions are 34.2 m/s and 19.6 m/s, respectively. As expected, the dominate flow directions are top to bottom on the right-hand side of the turbine, and right to left along the bottom of the turbine.

Figure 30: Velocity vectors colored by x-direction magnitude where the velocity magnitudes are given in m/s.
6.5. PRESSURE

Figures 32 and 33 depict the contours of absolute pressure in the region of symmetry. Figure 32 displays the entire range of absolute pressures present in the turbine; while Figure 33 presents only the pressures above atmospheric pressure. Figure 32 essentially says the bulk of the fluid in the turbine is at atmospheric pressure with only small deviations. Thus Figure 33 says the WCFT is behaving as an impulse turbine. Figure 33 shows that there are some small pressure deviations across the rotor, but they are very small. Since the minimum pressure in Figure 33 was set to atmospheric, the large white regions in Figure 33 either represent a solid structure (turbine blade) or a fluid region with a pressure slightly below atmospheric.

Operating conditions for the simulations were set to standard atmospheric temperature and pressure. Although temperature is not simulated, pressure is tracked in a detailed manner. Only minor fluctuations in pressure are noticed. Pressure at the upper exit portion of the turbine casing and in regions around a couple of the blades are lower than atmospheric pressure. These three low pressure regions are associated with higher velocities.
Figure 34 shows, in a more precise manner, that the pressure deviations in the turbine are small. Figure 34 is a histogram of the absolute pressures in the entire region of fluid flow. Over 70% of the flow volume is in the pressure range from 102 to 103 kPa. Thus the bulk of the turbine chamber is slightly above atmospheric pressure. About 25% of the total volume lies in the region where the pressure is less than atmospheric. The minimum absolute pressure in the turbine is 97.8 kPa; while the maximum pressure recorded is 104 kPa. Most of the volume within the turbine casing is essentially at atmospheric pressure.

Figure 32: Contours of absolute pressure
Figure 33: Contours of absolute pressure above atmospheric pressure.

Figure 34: Pressure histogram.
6.3. POWER VERSUS ROTATIONAL SPEED

In this study, what is desired is the peak shaft power that can be delivered by the WCFT. For this reason plots of turbine performance are made as a function of turbine rotational speed. To prepare these plots, time consuming CFD simulations had to be done at several turbine rotational speeds. Each turbine rotational speed required 10 to 40 hours of supercomputer CPU time. In this work these plots are made for both a 12 bladed WCFT and a 9 bladed WCFT to see which blade configuration delivers the best results. The WCFT studied experimentally was a 12-bladed turbine and it was desired to see the effect of reducing the number of blades; thus, a 9-bladed turbine is simulated. Both the twelve and nine bladed turbines use the base case conditions shown in Table 3, except for the number of blades and the rotational speed of the rotor is varied. In particular, the water mass flow rate into the turbine is the same in both cases.

6.6.1. Twelve Bladed Turbine

Figure 35 shows the average power produced by a twelve bladed WCFT as a function of the rotor rotational speed. Rotational speeds from 5 to 270 rpms were surveyed. All the powers in this curve were calculated using the torque on the turbine shaft times the rotor rotational speed (see Equation 6.3). As can be seen from Figure 35 this is essentially the entire operating range of this turbine. Of course, a zero-rotational speed could have been simulated, but this would provide zero power. Rotational speeds a great deal higher than 270 rpms could have been simulated as well; but these would have required power be input to the turbine. Figure 35 shows that peak turbine power is 9.01 watts at a rotational speed of 150 rpm, but the operating range from 115 to 185 rpm provides very good performance. There is a flat region on the power versus rotational speed curve. The general parabolic shape of the power versus rotational speed curve is typical of WCFTs.
As mentioned in the objectives presented in Chapter 1, it is desired to make a recommendation of how to improve the performance of the current WCFT being used in the Water Resource Lab at Central State University. This is the reason for analyzing a WCFT with a different number of blades. The different number of blades chosen was nine. This is a reduction of three blades from the current design of twelve blades. In this subsection, the results for the nine-bladed turbine are presented; and in the next subsection comparisons between the twelve and nine bladed turbines are made.

Figure 36 shows the average power output of a nine bladed WCFT as a function of the rotor rotational speed. The same range of rotational speeds as tested with the twelve-bladed turbine is covered here. Again, all the powers in this curve were calculated using average turbine shaft torque times the rotational speed, just like the twelve-bladed turbine. Figure 36 shows that peak turbine power is 9.52 watts at a rotational speed of 150 rpm, and the operating range from 100 to 190 rpm provides very good performance. Powers greater than 8.6 watts are considered good performance. Once again, a general parabolic shape is obtained.
6.6.3. Comparison of Twelve Bladed Turbine to a Nine Bladed Turbine

A plotted comparison of the performance of a twelve bladed WCFT and a nine bladed is given in Figure 37. This plot clearly shows that the nine bladed WCFT outperforms the twelve bladed WCFT. The differences are not large, but they are noticeable. The noticeable differences come in the middle portion of the rotational speed range shown in Figure 37, while almost identical behavior is obtained at low and high speeds. The maximum power of the twelve-bladed turbine is 9.01 watts, while that for the nine-bladed turbine is 9.52 watts. The twelve-bladed turbine also has a narrower speed range above 8.6 watts, than the nine-bladed turbine.

Based on the results in Figure 37, it is concluded that the nine-blade turbine is more efficient than the twelve bladed turbine and is the turbine of choice for the operating conditions outlined in Table 3. While this is a very interesting finding, it must be remembered that the power curves shown in Figure 37 are for one water mass flow rate. If the mass flow rate over the dam changes, it may be better to use a twelve-bladed turbine. The current investigator believes that the optimum number of blades for a WCFT depends on the water mass flow rate and the velocity of the fluid entering the turbine. Higher velocity flows will probably require turbines with more
blades, while lower velocity flows will probably require turbines with less blades to obtain optimum performance.

The one thing that should be concluded from this work is that CFD simulations can provide a great deal of useful information on WCFT design. Of course, this design work can be done experimentally, but the cost and time is much less doing the bulk of it computationally. Once an optimum design configuration is obtained a few experiments can be done to see that the CFD analysis agrees with reality. This is opposed to doing a great many experiments to locate the optimum design for a given application. As a result of this thesis work, a CFD simulation tool for WCFTs has been developed. This tool can be used by a follow-on graduate student to survey a wide portion of the design space. This would be very helpful to this new turbine design which Central State has called the William’s cross flow turbine.

![Figure 37: Turbine power comparison between twelve and nine bladed WCFTs.](image)

73
CHAPTER 7
CONCLUSIONS AND RECOMMENDATIONS

7.1. CONCLUSIONS

Small hydropower electricity generation appears to be a substantial renewable energy source that society is not taking full advantage of tapping. It has been estimated that this source of renewable energy could be used to generate 347 TWh of electricity yearly in the United States. Around the world there appears to be many opportunities as well. Even more exciting is part of the infrastructure for tapping this energy source is already in place. It is estimated that there more than 80,000 small dams around the United States that are simply used for flood control, and no electric power generation is occurring. By installing low-head turbines along these dams, useful electric power can be generated. The work in this thesis is a study of one such low-head turbine that has just recently been designed in the Water Resource Lab at Central State University. This turbine is a variant of the long used cross flow turbine, and has been named the Williams cross flow turbine (WCFT) in honor of its inventor.

A major accomplishment of this thesis work is the development of a computational fluid dynamics model of a WCFT. This program was written using the commercial software ANSYS Fluent. The computational model is very comprehensive in that it uses a detailed geometric
representation of a WCFT, it is three-dimensional, it is unsteady, it is multiphase in that both liquid water and air are included, it performs a detailed numerical solution of the full, incompressible, Navier-Stokes equations, and it includes the effects of turbulence. A computational tool such as this will be useful in the tapping of the vast, low-head hydropower resource that exists in the United States and around the world.

To aid in the development of the computational model, a limited amount of experimental work has been undertaken on the WCFT. This experimental work was undertaken in the facilities located in the Water Resource Lab at Central State University. These facilities include a state-of-the-art water channel, a lab-scale WCFT, and appropriate diagnostic tools. Working in this lab provided knowledge and insight to the development of the computational model. It also provided some experimentally measured results; to which, results from the computational model could be compared.

A comparison between an average power obtained from the computational model and an average power obtained from the experiment was done as part of this thesis work. Experimental measurements produced an average turbine power output of 10.8 watts for a, twelve bladed, WCFT. The computation produced an average turbine power of 8.85 watts. This is a difference of 18.1% from the experimentally obtained results. Due to the complexities involved in an analysis such as this, an 18.1% difference is deemed reasonable.

To keep computational times low, it was desired to use as few grid points and as large a time step as possible, without compromising the results. Keeping the number of grid points and the number of time steps small was necessary in this work, because the computational times required are between 15 to 40 hours on a supercomputer using 12 processors. To demonstrate that the final results are not affected by the grid and time step chosen, grid and time independence studies have been undertaken. The grid study indicated that approximate 1.3 million grid points are required to obtain results within 2% of being completely independent of the grid for the model WCFT simulated in the validation run. Therefore, in all the results presented in this thesis, approximately 1.3 grid points are used. The time step study was not as conclusive as the grid study, but indicates that a time step of 0.0005 seconds provides results within 4% of convergence for the lab-scale WCFT. At time steps smaller than 0.0004 seconds, it appears that numerical inaccuracies
start to appear. For this reason, a time step of 0.0005 seconds was used to produce the final results presented in this thesis.

A goal of this work was to make some recommendation to improve the performance of the current lab-scale WCFT used in the Water Resource Lab at Central State University. This was done. The computational model developed as part of this work was used to simulate the performance of a twelve bladed, lab-scale WCFT and a nine bladed, lab-scale WCFT. To correctly determine which of these WCFT turbines performs better for a given water flow rate and water head, computational results were produced for many different turbine rotor rotational speeds. Changing the rotational speeds is a way to show the effect of the load placed on the turbine. To properly compare the performance of two different turbines for given input conditions, power versus rotor rotational speed curves were produced for each turbine. These curves indicate that the nine bladed WCFT performed better than the twelve bladed WCFT that was tested in the Central State Water Resource Lab. Both turbines were shown to produce powers over 8.5 watts for a range of rotor rotational speeds; however, the twelve-bladed turbine had a peak power of 9.01 watts and the nine bladed turbine has a peak power of 9.5 watts.

It must be realized that the comparison of the powers of the twelve-bladed turbine and the nine bladed turbine were done at one water flow rate and one water head condition. It may be that the final conclusion drawn from such a comparison will be different if the water flowrate and head are changed. While the limited comparison done between the twelve and nine blade turbines as part of this thesis work shows an interesting result, the bigger conclusion that should be drawn from this result is that there is much that can be done to improve the design of the lab-scale WCFT turbine and to design full-scale WCFTs. The computational fluid dynamics model developed as part of this work is the proper tool to do this.

To further understand the operation of a WCFT a large number of field results have been produced for the nine bladed WCFT. These field results include water volume fractions, a velocity vector plot colored to indicate velocity magnitude, a velocity vector plot colored to indicate fluid density, a velocity vector plot colored to indicate x-direction velocity magnitude, a velocity vector plot colored to indicate y-direction velocity magnitude, a velocity vector plot colored to indicate z-direction velocity magnitude, and fluid pressure plots. These plots are all taken at the symmetry cross sectional plane of the turbine, but results have been computed to produce these plots at any
cross-sectional area of the turbine. To further quantify some of the field results, histogram plots are given. Plots such as these are helpful in understanding the performance of a WCFT turbine. Velocity vector plots are especially helpful in determining the water flow pattern. The velocity vector plot colored to indicate fluid density presented in this thesis shows that much of the water flowing through the nine bladed WCFT does indeed strike the rotor blades on two sides of the rotor. This plot also shows that some of the water does not do this. Maybe the WCFT design can be improved to reduce the amount of water that only strikes the rotor blades on one side of the rotor.

7.2. RECOMMENDATIONS

It is believed that this thesis work has shown that the computational fluid dynamics model developed here can be helpful in improving the design of the current WCFT and for producing designs of future WCFTs for many different operating conditions. For this reason, the main recommendation made is to further the study of the WCFT using this computational model. Many different operating conditions should be studied, and many different sizes of turbines should be studied. This will take a considerable effort, but it will be beneficial to the future development of WCFTs.

While the developed computational model has been verified against an experimental result, more verification should be undertaken. Many more experimental operating conditions should be simulated and compared. Again, this will require considerable effort, but due to the benefits of such a model, the effort is worth it. To further verify the computational model, additional experimental measurements should be taken. It would be nice to get some sort of flow visualization inside the turbine. This could possibly be done by making the turbine casing out of clear plastic and making particle-image-velocity measurements. Just obtaining an eyeball look at the flow in the turbine would be beneficial as well.

On a policy level, it is recommended that more be done to further the utilization of low-head, hydropower across the United States and throughout the world. This seems to be a resource ready for the tapping. It is recommended that more resources be committed to getting these systems installed at some of the 80,000 sites currently available. Financial and tax incentives should be provided in order to encourage private industry to be involved in this effort. To further encourage private industry to be involved, federal and state laws regarding small hydropower development
should be clearly defined. To make small hydropower installations as efficient and profitable as possible, more funding should be committed to research on this topic.

Lastly, it is recommended that Central State University install a prototype WCFT in the Miami River in the Dayton, Ohio region. This will do much to advance the case for using WCFTs in low-head, hydropower facilities.
References


