COMPUTATION AND ANALYSIS OF CAVITATING FLOW IN
FRANCIS-CLASS HYDRAULIC TURBINES

A Dissertation in
Engineering Science and Mechanics
by
Daniel J. Leonard

© 2015 Daniel J. Leonard

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Doctor of Philosophy

May 2015
The dissertation of Daniel J. Leonard was reviewed and approved* by the following:

Scott T. Miller  
Research Associate, Applied Research Laboratory, and  
Assistant Professor of Engineering Science and Mechanics  
Dissertation Advisor, Chair of Committee

Jules W. Lindau  
Research Associate, Applied Research Laboratory, and  
Assistant Professor of Aerospace Engineering  
Special Member

Francesco Costanzo  
Professor of Engineering Science and Mechanics and Mathematics

Jonathan S. Pitt  
Research Associate, Applied Research Laboratory, and  
Assistant Professor of Engineering Science and Mechanics

John M. Cimbala  
Professor of Mechanical Engineering

Brent A. Craven  
Adjunct Professor of Mechanical Engineering

Judith A. Todd  
Professor of Engineering Science and Mechanics  
Head of the Department of Engineering Science and Mechanics

*Signatures are on file in the Graduate School.
Abstract

Hydropower is the most proven renewable energy technology, supplying the world with 16% of its electricity. Conventional hydropower generates a vast majority of that percentage. Although a mature technology, hydroelectric generation shows great promise for expansion through new dams and plants in developing hydro countries. Moreover, in developed hydro countries, such as the United States, installing generating units in existing dams and the modern refurbishment of existing plants can greatly expand generating capabilities with little to no further impact on the environment. In addition, modern computational technology and fluid dynamics expertise has led to substantial improvements in modern turbine design and performance.

Cavitation has always presented a problem in hydroturbines, causing performance breakdown, erosion, damage, vibration, and noise. While modern turbines are usually designed to be cavitation-free at their best efficiency point, due to the variable demand of the energy market it is fairly common to operate at off-design conditions. Here, cavitation and its deleterious effects are unavoidable, and hence, cavitation is a limiting factor on the design and operation of these turbines. Multiphase Computational Fluid Dynamics (CFD) has been used in recent years to model cavitating flow for a large range of problems, including turbomachinery. However, CFD of cavitating flow in hydroturbines is still in its infancy.

This dissertation presents steady-periodic Reynolds-averaged Navier-Stokes simulations of a cavitating Francis-class hydroturbine at model and prototype scales. Computational results of the reduced-scale model and full-scale prototype, undergoing performance breakdown, are compared with empirical model data and prototype performance estimations based on standard industry scalings from the model data. Mesh convergence of the simulations is also displayed. Comparisons are made between the scales to display that cavitation performance breakdown can occur more abruptly in the model than the prototype, due to lack of Froude similitude between the two. When severe cavitation occurs, clear differences are observed in vapor content between the scales. A stage-by-stage performance decomposition is conducted to analyze the losses within individual components of each scale of the machine. As cavitation becomes more severe, the losses in the draft tube account...
for an increasing amount of the total losses in the machine. More losses occur in the model draft tube as cavitation formation in the prototype draft tube is prevented by the larger hydrostatic pressure gradient across the machine.

Additionally, unsteady Detached Eddy Simulations of the fully-coupled cavitating hydroturbine are performed for both scales. Both mesh and temporal convergence studies are provided. The temporal and spectral content of fluctuations in torque and pressure are monitored and compared between single-phase, cavitating, model, and prototype cases. A shallow draft tube induced runner imbalance results in an asymmetric vapor distribution about the runner, leading to more extensive growth and collapse of vapor on any individual blade as it undergoes a revolution. Unique frequency components manifest and persist through the entire machine only when cavitation is present in the hub vortex. Large maximum pressure spikes, which result from vapor collapse, are observed on the blade surfaces in the multiphase simulations, and these may be a potential source of cavitation damage and erosion.

Multiphase CFD is shown to be an accurate and effective technique for simulating and analyzing cavitating flow in Francis-class hydraulic turbines. It is recommended that it be used as an industrial tool to supplement model cavitation experiments for all types of hydraulic turbines. Moreover, multiphase CFD can be equally effective as a research tool, to investigate mechanisms of cavitating hydraulic turbines that are not understood, and to uncover unique new phenomena which are currently unknown.
# Table of Contents

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

## Chapter 1  
**Introduction** 1  
1.1 Hydropower ........................................... 1  
1.1.1 Hydroturbines ........................................ 4  
1.2 Cavitation ............................................ 11  
1.2.1 Turbomachinery Cavitation .......................... 14  
1.2.2 Hydroturbine Cavitation ............................ 15  
1.3 Computational Fluid Dynamics .......................... 19  
1.3.1 CFD Applied to Cavitating Flow .................... 21  
1.4 Scope of Contributions ................................. 32  
1.4.1 Layout of Dissertation ............................. 34  

## Chapter 2  
**Steady-Periodic Methods and Approach** 35  
2.1 Overview ............................................ 35  
2.1.1 Equations of Motion for a Mixture .................. 35  
2.2 Computational Approach to Cavitating Flow ............ 36  
2.2.1 Computational Approach to Multi-stage Hydroturbines ... 38  
2.2.2 Mass Transfer Modeling ............................. 42  
2.2.3 Turbulence Modeling ............................... 43  
2.2.4 Interface/Shock-Capturing ........................... 44  
2.2.5 Spatial Discretization .............................. 44  
2.2.6 Steady-Periodic Solution Strategy ................. 48
2.3 Details of Francis Hydroturbine Case .......................... 49
  2.3.1 Geometry ................................................. 49
  2.3.2 Computational Mesh ....................................... 51
  2.3.3 Boundary Conditions ...................................... 54
  2.3.4 Initial Conditions and Convergence ....................... 56
  2.3.5 Computational Resources ................................. 57
2.4 Model Experiments ............................................. 57

Chapter 3
Steady-Periodic Results and Discussion 62
3.1 Introduction .................................................. 62
3.2 Single-Phase Flow ............................................ 64
  3.2.1 Mesh Convergence Study ................................. 64
  3.2.2 Single-phase Results ..................................... 66
3.3 Cavitating Machine Performance Results ...................... 69
  3.3.1 Mesh Convergence Study ................................... 70
  3.3.2 Multiphase Machine Performance Results ................ 74
3.4 Stage-by-Stage Performance Decomposition .................... 95
3.5 Summary ...................................................... 108

Chapter 4
Unsteady Methods and Approach 110
4.1 Overview ..................................................... 110
4.2 Segregated Approach to CFD ................................ 110
  4.2.1 Foundations of Projection Methods ...................... 111
  4.2.2 Governing Equations and Segregated Procedure ......... 112
  4.2.3 Volume of Fluid Method .................................. 119
  4.2.4 Detached Eddy Simulation ................................ 120
  4.2.5 Cavitation Modeling ...................................... 122
  4.2.6 Unsteady Solution Strategy ............................... 123
4.3 Fully-Coupled Francis Hydroturbine Case ....................... 124
  4.3.1 Geometry and New Mesh Components ..................... 124
  4.3.2 Boundary Conditions .................................... 127
  4.3.3 Initial Conditions and Convergence ..................... 128
  4.3.4 Computational Resources ................................. 129
4.4 Unsteady Model Experiments .................................... 129

Chapter 5
Unsteady Results and Discussion 131
5.1 Introduction .................................................. 131
5.2 Convergence ............................................. 133
  5.2.1 Time-averaged Variable Convergence .............. 134
  5.2.2 Unsteady Variable Convergence .................... 137
5.3 Unsteady Results ....................................... 140
  5.3.1 Time-Averaged Unsteady ............................ 141
  5.3.2 Precessing Vortex Core ............................ 148
  5.3.3 Unsteady Torque on Runner Blades ................ 149
  5.3.4 Pressure Fluctuations .............................. 157
  5.3.5 Maximum Pressure on Runner Blade Surface .......... 164
5.4 Commentary on Potential Cavitation Erosion ............ 167
5.5 Summary ................................................ 169

Chapter 6
  Summary and Conclusions ................................. 171
  6.1 Overview ............................................... 171
  6.2 Contributions and Findings ........................... 172
  6.3 Future Work .......................................... 173
  6.4 Conclusion ............................................ 174

Appendix A
  Spalart-Allmaras DDES Model ............................. 175

Appendix B
  Non-Technical Abstract .................................. 179

Bibliography .................................................. 181
List of Figures

1.1 Drawing of a general hydroelectric plant. . . . . . . . . . . . . . . . . 3
1.2 Drawings of the two primary classes of reaction turbines. . . . . . . 5
1.3 Common components of reaction hydroturbines. . . . . . . . . . . . . 7
1.4 Photos of the primary types of large-scale cavitation. . . . . . . . . 13
1.5 Drawing of a typical reentrant jet. . . . . . . . . . . . . . . . . . . . 14
1.6 Typical cavitation damage at the discharge of a Francis runner. . . 16
1.7 A hill chart of a Francis turbine showing cavitation limits. . . . . . 16
1.8 Efficiency breakdown of a Francis turbine. . . . . . . . . . . . . . . 17
1.9 Photos of hub vortex cavitation in a model test. . . . . . . . . . . . . 18
1.10 Damage to a Francis turbine from cavitating interblade vortices. . 18
1.11 Mixture sound speed versus vapor volume fraction. . . . . . . . . . 22
1.12 Cavitation breakdown of experiment and various cavitation models. 25
1.13 Plot/images of pressure/cavity volume fluctuations in draft tube. . 27
1.14 Pressure pulsations and cavitation in runner and draft tube. . . . . 28
1.15 Reduction in vapor content on runner blade due to perforation. . 31
2.1 Periodicity of guide vane and runner blade stages. . . . . . . . . . . 41
2.2 Geometry of the Francis turbine under computational investigation. 50
2.3 Fine mesh of the guide vane. . . . . . . . . . . . . . . . . . . . . . . 53
2.4 Fine mesh of the shroud side of the runner blade. . . . . . . . . . . 53
2.5 Fine mesh of the draft tube and hub nose cone. . . . . . . . . . . . . 54
2.6 Periodic and mixing plane boundaries. . . . . . . . . . . . . . . . . 56
2.7 Inlet and outlet of generic run-of-river plant. . . . . . . . . . . . . . 59
2.8 Efficiency breakdown of model experiment and prototype estimates. 61
3.1 Efficiency mesh convergence for single-phase flow. . . . . . . . . . . 66
3.2 Contours of static pressure in single-phase turbine. . . . . . . . . . 67
3.3 Contours of hydrostatic pressure in single-phase draft tube. . . . . 68
3.4 Contours of static pressure in single-phase draft tube. . . . . . . . . 68
3.5 Streamlines in single-phase draft tube. . . . . . . . . . . . . . . . . 69
3.6 Contours of vertical velocity in single-phase draft tube. . . . . . . . 69
3.7 Efficiency mesh convergence for multiphase flow. ............... 72
3.8 Percentage of vapor mesh convergence. .......................... 73
3.9 Effect of Thoma number on net specific energy coefficient. .... 76
3.10 Effect of Thoma number on flow coefficient. ................. 78
3.11 Effect of Thoma number on power coefficient. ............... 80
3.12 Effect of Thoma number on machine efficiency. .............. 82
3.13 Model isosurfaces of vapor volume fraction for Case 1 \( (\sigma = 1.24) \). 84
3.14 Cavitation drawings and photo from Case 1 \( (\sigma = 1.24) \) model tests. 84
3.15 Model isosurfaces of vapor volume fraction for Case 2 \( (\sigma = 0.99) \). 85
3.16 Cavitation drawings and photo from Case 2 \( (\sigma = 0.99) \) model tests. 85
3.17 Model isosurfaces of vapor volume fraction for Case 3 \( (\sigma = 0.70) \). 86
3.18 Cavitation drawings and photo from Case 3 \( (\sigma = 0.70) \) model tests. 86
3.19 Model isosurfaces of vapor volume fraction for Case 4 \( (\sigma = 0.52) \). 87
3.20 Cavitation drawings and photo from Case 4 \( (\sigma = 0.52) \) model tests. 87
3.21 Model isosurfaces of vapor volume fraction for Case 5 \( (\sigma = 0.34) \). 88
3.22 Model isosurfaces of vapor volume fraction for Case 6 \( (\sigma = 0.20) \). 89
3.23 Model isosurfaces of vapor volume fraction for Case 7 \( (\sigma = 0.13) \). 90
3.24 Prototype isosurfaces of vapor volume fraction for Case 1 \( (\sigma = 1.24) \). 91
3.25 Prototype isosurfaces of vapor volume fraction for Case 2 \( (\sigma = 0.99) \). 92
3.26 Prototype isosurfaces of vapor volume fraction for Case 3 \( (\sigma = 0.70) \). 92
3.27 Prototype isosurfaces of vapor volume fraction for Case 4 \( (\sigma = 0.52) \). 93
3.28 Prototype isosurfaces of vapor volume fraction for Case 5 \( (\sigma = 0.34) \). 93
3.29 Prototype isosurfaces of vapor volume fraction for Case 6 \( (\sigma = 0.20) \). 94
3.30 Prototype isosurfaces of vapor volume fraction for Case 7 \( (\sigma = 0.13) \). 94
3.31 Surfaces for calculating flux of mechanical energy. ........... 96
3.32 Effect of \( \sigma \) on true machine efficiency. .......................... 97
3.33 Effect of \( \sigma \) on runner-only efficiency. .......................... 98
3.34 Effect of \( \sigma \) on kinetic energy recovery coefficient. .......... 100
3.35 Effect of \( \sigma \) on potential energy recovery coefficient. ........ 101
3.36 Effect of \( \sigma \) on loss coefficient. ................................. 103
3.37 Effect of \( \sigma \) on kinetic-to-potential energy conversion ratio. .... 104
3.38 Effect of \( \sigma \) on percent of power lost in model-scale stages. ... 106
3.39 Effect of \( \sigma \) on percent of power lost in prototype-scale stages. ... 107
3.40 Effect of \( \sigma \) on dimensional power lost in model/prototype stages. 108
4.1 Complete geometry of the Francis turbine under investigation. ... 125
4.2 Mesh of the penstock, partial spiral casing, and guide vanes. .... 126
4.3 Meshes of the penstock and stay vane regions. ................. 126
5.1 Locations of pressure probes 1 and 2 in the draft tube. .......... 132
5.2 Surfaces depicting pressure probes A and B in penstock. 
5.3 Convergence of time-averaged machine efficiency. 
5.4 Convergence of time-averaged $V_v$ in computational domain. 
5.5 Convergence of normalized torque fluctuations on blade-1. 
5.6 Convergence of normalized pressure fluctuations at probe-1. 
5.7 Images of vapor content beneath the hub for various mesh levels. 
5.8 Contour plots of $d - \Psi C_{DES} \Delta \geq 0$ and $\mu_T$. 
5.9 Effect of Thoma number on machine efficiency. 
5.10 Model isosurfaces of vapor volume fraction for Case 2 ($\sigma = 0.99$). 
5.11 Cavitation drawings and photo from Case 2 ($\sigma = 0.99$) model tests. 
5.12 Prototype isosurfaces of vapor volume fraction for Case 2 ($\sigma = 0.99$). 
5.13 Model isosurfaces of vapor volume fraction for Case 3 ($\sigma = 0.70$). 
5.14 Cavitation drawings and photo from Case 3 ($\sigma = 0.70$) model tests. 
5.15 Prototype isosurfaces of vapor volume fraction for Case 3 ($\sigma = 0.70$). 
5.16 Piezometric pressure contours at the exit surface of the runner. 
5.17 Prototype $\sigma = 0.99$ with $\alpha_v = 0.2$ at 3.1059 rev. 
5.18 Snapshot of precessing vortex cores from single-phase CFD. 
5.19 Time-series of normalized torque fluctuations for blade-1. 
5.20 Amplitude spectra of normalized torque fluctuations on blade-1. 
5.21 Vapor collapse on prototype runner blade surface for Case 2 ($\sigma = 0.99$). 
5.22 Vapor collapse on prototype runner blade surface for Case 3 ($\sigma = 0.70$). 
5.23 Amplitude spectra of $T_E$ on entire runner. 
5.24 Amplitude spectra of $T_E$ on entire runner scaled by log$_{10}$. 
5.25 Time-series of normalized pressure fluctuations at probe-1. 
5.26 Amplitude spectra of $p_E$. 
5.27 Amplitude spectra of $p_E$ at probe-1 scaled by log$_{10}$. 
5.28 Amplitude spectra of normalized volume of vapor fluctuations. 
5.29 Shedding and collapse of hub vortex vapor in model Case 2 ($\sigma = 0.99$). 
5.30 Time-series of normalized maximum pressure on blade-1. 
5.31 Amplitude spectra of normalized maximum pressure on blade-1. 
5.32 Physical process of cavitation-induced solid surface pitting.
List of Tables

2.1 Number of cells in each periodic stage of coarse and fine meshes. . . . 52
2.2 Values of $y^+$ for each stage of model and prototype. . . . . . . . . 56

3.1 Efficiency mesh convergence for single-phase flow. . . . . . . . . . . 66
3.2 Efficiency mesh convergence for multiphase flow. . . . . . . . . . . . 71
3.3 Percentage of vapor mesh convergence. . . . . . . . . . . . . . . . . . 71
3.4 Net specific energy coefficients of all cases. . . . . . . . . . . . . . . . 75
3.5 Flow coefficients of all cases. . . . . . . . . . . . . . . . . . . . . . . 77
3.6 Power coefficients of all cases. . . . . . . . . . . . . . . . . . . . . . 79
3.7 Efficiency of all cases. . . . . . . . . . . . . . . . . . . . . . . . . . . . 81
3.8 Percent differences in efficiency between model/prototype scales. . . 81

4.1 Number of cells in each stage for coarse, medium, and fine meshes. . 127
4.2 Three $\Delta t$ values and number of time-steps per revolution. . . . . . 128

5.1 Maximum convective and mesh Courant numbers. . . . . . . . . . . . 135
5.2 Peak-to-peak amplitudes of normalized torque for blade-1. . . . . . . 151
5.3 Peak-to-peak amplitudes of normalized pressure for all probes. . . . . 159
List of Symbols

$C_{KR}, C_{PR}$  Kinetic and potential energy recovery coefficients

$Co$  Courant number

$d$  Distance from nearest wall

$\tilde{d}$  Turbulent length scale

$D$  Runner diameter at runner exit

$E$  Net specific energy

$E$  Flux vector

$Fr$  Froude number

$g, g_i, g$  Gravitational acceleration vector and scalar component(s)

$h_{cell}$  Representative cell size

$H$  Source vector

$\dot{m}, \dot{m}^+, \dot{m}^-$  Mass transfer between phases

$\dot{M}_f$  Mass flux at cell face

$n$  Unit normal vector

$NPSE$  Net positive specific suction energy

$p, p_v$  Pressure and vapor pressure

$p_E$  Normalized pressure fluctuations

$P, P^*$  Power and power coefficient
\( Q \) Volumetric flow rate

\( Q_e, Q \) Primitive and pseudo-time variable vectors

\( R \) Runner radius at runner exit

\( \mathcal{R} \) Spherical bubble radius

\( Re \) Reynolds number

\( S \) Local surface area

\( t, \Delta t \) Time and time-step size

\( T_E \) Normalized torque fluctuations

\( \mathcal{T} \) Torque

\( v, v_i, v \) Velocity vector and scalar component(s)

\( V \) Local volume

\( x, x_i, x \) Position vector and scalar component(s)

\( \Delta x \) Local cell spacing

\( y^+ \) Dimensionless distance of cell center from wall

\( Z \) Altitude

**Greek Symbols**

\( \alpha_v, \alpha_l \) Vapor and liquid volume fractions

\( \Phi^p \) Preconditioning matrix

\( \Delta \) Grid filter length scale for DES

\( \Delta \Xi \) Power lost (not used to do work on runner)

\( \epsilon \) Error

\( \zeta \) Loss coefficient

\( \eta \) Efficiency
\( \mu, \mu_v, \mu_l \) Mixture, vapor, and liquid dynamic viscosities

\( \nu, \tilde{\nu} \) Mixture kinematic viscosity and modified diffusivity

\( \rho, \rho_v, \rho_l \) Mixture, vapor, and liquid densities

\( \sigma \) Thoma number

\( \tau \) Pseudo-time

\( \boldsymbol{\tau}, \tau_{ij} \) Mean viscous stress tensor and scalar components

\( \phi \) Flow coefficient

\( \varphi \) Arbitrary scalar variable

\( \chi \) Kinetic-to-potential energy conversion ratio

\( \psi \) Net specific energy coefficient

\( \omega \) Angular frequency

\( \Omega \) Rotation rate of runner

**Subscripts**

\( abs \) Absolute

\( l \) Liquid

\( M \) Model

\( P \) Prototype

\( p - p \) Peak-to-peak amplitude

\( rel \) Relative

\( sta \) Standard

\( T \) Turbulent

\( v \) Vapor
∞ Freestream

Abbreviations

BC  Boundary Condition
BEP Best Efficiency Point
CD  Central Differencing
CFD Computational Fluid Dynamics
CV  Control Volume
DES, DDES Detached Eddy Simulation, Delayed DES
EXP Experimental
FFT, fft Fast Fourier Transform
LES Large Eddy Simulation
MUSCL Monotonic Upstream-Centered Scheme for Conservation Laws
N-S Navier-Stokes
PVC Precessing Vortex Core
RANS, URANS Reynolds Averaged Navier-Stokes, Unsteady RANS
R-P Rayleigh-Plesset
SOUBCD Second Order Upwind Bounded Central Differencing
SIMPLE Semi-Implicit Method for Pressure Linked Equations
TVD Total Variation Diminishing
UD Upwind Differencing
VOF Volume of Fluid
WAH Weir American Hydro
Acknowledgments

I would like to thank my research advisor, Dr. Jules Lindau, who played an integral role in my understanding of all aspects of the work presented in this dissertation. He was always available to answer my questions and provide guidance, and I am appreciative of his patience with me over the years. Additionally, I am grateful for the help that my committee has provided me in the final stages of my degree.

The information, data, or work presented herein was funded in part by the Office of Energy Efficiency and Renewable Energy (EERE), U.S. Department of Energy, under Award Numbers DE-EE0002667 (DOE/PSU Graduate Student Fellowship Program for Hydropower Research) and DE-EE0002668 (HRF Fellowship) and the Hydro Research Foundation. I am thankful to Dr. John Cimbala for coordinating the PSU Hydropower Research Group, as well as to Brenna Vaughn and Deborah Linke for coordinating the HRF fellowship. These fellowships allowed me to meet and make friends with students from across the country, that were working on a range of hydropower topics. Also, they provided opportunities to interact and collaborate with industry partners, such as Weir-American Hydro, who provided important experimental contributions and expertise to this work. In particular, I would like to thank Joe Hill, my industry sponsor for the HRF Fellowship, as he provided guidance on the experimental results to which the computations in this dissertation were compared to.

Over the course of my time as a graduate student, I had the privilege to conduct my research in the Garfield Thomas Water Tunnel Building of the Applied Research Laboratory. In this environment, I was able to interact with employees and students who focused on fluid dynamics research, while being inspired by the large range of research possibilities in the field. This was truly a great benefit to my education. Many of the students here have become my good friends, and they helped make long hours in the lab more enjoyable.

Most importantly, I would like to express my deepest gratitude to my family and close friends. Without their love and support, I probably would have never even dreamed of such an accomplishment. In particular, I am indebted to my parents, who have believed in me since day one.
1 | Introduction

1.1 Hydropower

Hydropower is the harnessing of energy from Earth’s hydrological cycle for the generation of power. The hydrological cycle is driven by solar radiation, through the evaporation of water, primarily from Earth’s oceans. This vapor then travels through the atmosphere, and about one-fifth of it is deposited on land in its liquid phase, by means of precipitation (Kumar et al., 2011). The remaining falls over the oceans. Water which falls on land is then driven back towards the oceans by gravitational forces. Some of the energy in the flow of river water can be extracted, and it is this portion of the energy contained in the hydrological cycle which can be used to generate hydropower.

Historically, hydropower has been used by civilizations for thousands of years. From the use of water wheels for the direct operation of grinding stones in flour mills, to the irrigation of land for agricultural purposes, people have tapped into this abundant resource of energy. However, a revolution in technology came about in the late 19th century, when an electric generator was driven by a hydroturbine on Fox River in Wisconsin (USBR, 2009). The technology of hydroelectric power was born.

Hydroelectric power is a clean, renewable, and highly efficient source of electricity generation, with an enormous potential for improvement and expansion. Hydroplant design is site specific because the flow resources at any location are the main factors determining how, as well as how much, energy can be extracted from the water. The two main flow resource parameters for a potential plant’s energy producing capacity are the river’s hydrostatic head and its flow rate. The variability of sites has resulted in plants which generate less than 5 kW of power to the 22.5
GW of Three Gorges Dam in China. This specificity provides unique challenges to engineers in each hydropower project.

Hydropower is the most proven renewable energy technology, supplying the world with 16 percent of its electricity, and some regions such as Quebec, Canada with 99 percent of their electricity. It has small operating costs, since the fuel for power generation is the water itself, and is free of pollution while operating. Another advantage is that the water remains available for many other uses, before and after generation. There are disadvantages as well, which have been mentioned by Raabe (1985) and summarized by Dixon (2005). There are large initial costs for development of a plant, a limited number of favorable sites, environmental concerns with dams, and most importantly to this present research, cavitation problems in the machines.

The industry still shows great promise for expansion even though the technology is so mature. The potential for hydropower generation is about four times of what is currently installed. In many cases, this potential exists in developing hydro countries such as China, India, and Brazil, but developed hydro countries, such as the U.S., still have the ability to greatly expand and update their hydro technology as well. Even though the U.S. plans to build no new dams in the foreseeable future, it still would like to double hydropower generation by 2030 (NHA, 2014). This can be accomplished by installing hydroelectric units in dams which do not generate hydropower, refurbishing inefficient old plants, and by upgrading antiquated turbines with state-of-the-art units developed with modern computational technology and fluid dynamics knowledge. Weir American Hydro (WAH) is a well established company that is in the business of increasing efficiency and power of North American hydroturbines by installing modern designs. WAH were the industry partners for the research presented in this dissertation. They provided geometry, knowledge, and experimental results of the hydroturbine under investigation, as well as guidance through their extensive knowledge base of the industry.

Conventional hydropower plants can be classified into three types of facilities. These are run-of-river, storage, and pumped storage plants, all of which are described in detail by Kumar et al. (2011). In short, run-of-river hydro plants do not store river water in a reservoir, but allow it to flow through the generating units as it would flow through a river; Storage plants have a reservoir to allow for more
adaptation to electricity demands; Pumped storage plants have a reservoir but can also operate in reverse, pumping water back into the reservoir, to be stored for when demand is higher. While run-of-river plants must deal with the variability of flow conditions on generation, which storage plants are not as susceptible to, the increased control of water by a storage plant results in a greater environmental impact than that caused by run-of-river plants.

With either the run-of-river or storage classifications, there are common components of a conventional hydroelectric plant. Figure 1.1 shows these components. A dam is present to create the head difference between the headrace (the water surface elevation upstream of the dam) and the tailrace (the water surface elevation downstream of the dam). The penstock comprises the pipes or tunnels which direct flow to the turbine. The hydroturbine itself is a mechanical device whose rotation is driven by the extraction of energy from the flow. The electric generator converts mechanical energy to electrical energy, and its rotation is driven by a shaft attached directly to the rotating hydroturbine. Finally, the draft tube is a diffuser which collects the flow after it exits the turbine, and deposits it on the lower side of the dam.
1.1.1 Hydroturbines

This dissertation focuses on the components of the hydroelectric plant which are responsible for guiding the water through the dam and extracting mechanical energy from the flow. There are two types of conventional hydraulic turbines, and these are reaction turbines and impulse turbines. The only impulse turbine that is used in modern hydropower is the Pelton turbine. This machine's specialization is for plants with extremely high head and low flow rate, which are most commonly storage plants. It operates by directing water through nozzles of small cross sectional area. The water exits the nozzles as high velocity jets, into ambient air, and these jets impinge upon bucket shaped blades placed around a wheel. The impulse of the jet imparts momentum to the turbine and rotates the wheel about its axis. Pelton turbines do not have much of a cavitation problem, and thus, the rest of this work is dedicated to reaction machines, in which cavitation is a much more significant drawback.

Unlike the Pelton turbine, all parts of a reaction machine are fully wetted by the flow. As mentioned by Dixon (2005), in a reaction turbine only a portion of the overall pressure drop has occurred when the flow reaches the turbine blades. The remaining pressure drop takes place as the turbine blades extract energy from the flow. There are two primary classes of reaction turbines. They are Francis turbines and Kaplan turbines. A Francis turbine, seen in Figure 1.2, is a turbine which has a radial inlet and an axial outlet, and is therefore usually categorized as a mixed-type turbine. Francis turbines can operate well over a wide range of flow conditions and are installed in run-of-river or storage plants. They are efficient at sites with fairly large head and flow, but also do well with fairly high flow and low head. They are usually not specialized enough to efficiently handle extremely high or low head locations, but they are the most common turbine employed at the highest capacity plants in the world, such as Three Gorges. The Kaplan turbine, seen in Figure 1.2 has an axial inlet into the turbine blades and axial outlet from the turbine blades, and is therefore categorized as an axial-type turbine. They specialize in many run-of-river plants that have low head and extremely high flow rate. The Kaplan looks similar to a propeller on a ship but run in reverse. Some of the unique features of the Kaplan are the tip clearances between the tips of the rotating blades and the machine walls, and that they sometimes have variable pitch.
blades which can be adjusted to flow conditions.

Both reaction and impulse turbines must rotate at a constant angular speed due to the constraints of the local electric power grid’s alternating current utility frequency. In most of the world, this frequency is 50 Hz but in some locations, such as North America, it is 60 Hz. Since the hydroturbine is directly connected to the electric generator, via a shaft, the turbine must rotate at a fixed frequency to produce electricity. Now, this does not mean that the turbine has to rotate at the electric utility frequency. The electric generator side of the machine includes rotor and stator components, with a particular distribution, and the electric generating frequency is determined by their interaction. Thus, the hydroturbine can rotate at a different angular speed than the utility frequency, but that speed must always remain fixed to ensure the electric generating frequency matches the utility frequency.

There are common stages amongst all reaction hydroturbines. Each of these stages will now be described in detail.

- **Penstock and Spiral Casing** The penstock is the piping or tunneling that delivers the flow to the turbomachine from the higher elevation side of the dam. The spiral casing is the piping which encircles the machine. It receives the flow from the penstock, and turns it in a manner to direct the flow at a certain angle into the stay vanes. Other than generating pre-swirl into the machine, the swirl collector reduces its cross-sectional area as it encircles the turbine. This design feature is to hopefully allow the same flow rate and flow angles at each circumferential position into the turbine. See Figure 1.3.

- **Stay Vanes** These are primarily stationary structural supports for the turbine
housing, located downstream of the swirl collector. Historically, many of these vanes were simple cylindrical struts. The bluff cylindrical shape can cause severe unsteadiness in the machine, as strong vortices shed from the body and impinge upon components further downstream (Dörfler et al., 2013). To alleviate this problem, the stay vanes of modern machines, and upgraded old machines, have been modified to a more hydrodynamic shape, commonly appearing as a ring of stationary hydrofoil. See Figure 1.3.

• **Guide Vanes** These vanes, located just downstream of the stay vanes, are pivotable, each vane about its own axis, to allow for flow control into the turbine. Guide vanes, also known as wicket gates, are essential features of reaction hydroturbines. They can be completely closed to shut off flow, or can be opened to different angles to allow varying flow rate, and flow angles, into the turbine. They are just upstream of the runner (the rotating component of the turbine which extracts energy from the flow), and thus, are vital in setting the flow conditions which the runner will encounter, and the efficiency of the machine. The guide vane angle is the most critical setting for the operation of a machine at a particular design point. The Kaplan, with variable pitch runner blades, still enjoys some additional freedom to adjust its design point at some particular guide vane angle. See Figure 1.3.

• **Runner** The runner is a rotating component of the turbine. The rotation is driven by the passing swirling flow, which does work on the runner blades through the transfer of angular momentum. This angular momentum transfer, from the flow to the runner, is the mechanism by which hydropower is generated in reaction machines. Francis runner blades are fixed on both ends. One end is fixed to the hub (also called the crown), and the other to the shroud (also called the band). Hence, the hub and shroud are also part of the rotating runner. The Kaplan turbine is fixed to the hub as well, but the other ends of the blades are free. The free blade tips make a small gap with the non-rotating shroud. When the flow exits a Francis or Kaplan runner, it flows into the draft tube. See Figure 1.3.

• **Draft Tube** This stationary component is responsible for delivering the flow from the runner exit to the tailrace, but also acts as a diffuser to ensure static pressure recovery. Its diffuser role is important to the performance of
the machine. Poor draft tubes can result in excessive cavitation, vibration, damage, and low efficiency turbines. Draft tubes usually have an elbow, changing the axial flow direction towards the tailrace. See Figure 1.3

Flow Features

Hydroturbines have many flow features which can be important to their operation, performance, and sustainability. Some of these features can also be responsible for causing cavitation to occur. Here, a few of these flow attributes are reviewed. This dissertation is focused on Francis turbines in particular, and most of the discussion is dedicated to flow through that class of turbine.

Swirling flow is an integral part of all reaction turbines. In fact, for steady and incompressible flow, the torque produced on the runner is equivalent to the flux of absolute angular momentum through the runner stage, and thus proportional to the difference in circumferential velocity (or swirl) through this stage (Wilcox, 2000). Swirl is commonly introduced into the flow by the spiral casing, adjusted by the guide vanes to provide precise flow angles into the runner at a specific operating condition, and extracted by the runner as torque on the blades.

Ideally, at the Best Efficiency Point (BEP) of the machine, all swirl has been extracted from the flow by the time it exits the runner stage, although it has proven beneficial to allow for small amounts of residual swirl entering into the draft tube. The kinetic energy due to residual swirl which exits the runner can not be recovered to static pressure in the draft tube, and therefore residual swirl will always cause
losses in the machine (Susan-Resiga et al., 2011). However, modern Francis turbine design recommends allowing for small amounts of residual swirl to enter into the draft tube. This allows for an improvement of the flow field within the draft tube by way of a well documented mechanism reported by Fox et al. (1971). In that study, conical diffusers with attached flow displayed minor performance benefits from swirling flow, however, when the flow was detached from the walls of the diffuser, the addition of swirl reattached the boundary layer, and substantially improved the performance. Thus, allowing small amounts of residual swirl in the draft tube can keep the boundary layer attached to the wall, which results in an overall performance gain even though losses can occur due to lack of pressure recovery.

Francis runners have become extremely efficient, providing optimum runner-only efficiencies of around 95% (Dixon, 2005) and thus have little hydraulic loss. Furthermore, numerical simulations of modern Francis turbines have shown that the spiral casing, vanes, and runner, all display smooth and moderate variations in hydraulic losses across the operating range (Vu and Retieb, 2002). However, at even slightly off-design conditions, the draft tube can display sharp increases in hydraulic losses, reducing the total efficiency of the machine considerably. As previously mentioned, the runner maintains reasonable losses in the draft tube at BEP by providing a small amount of residual swirl, but at off-design conditions the residual swirl exiting the runner will vary (since Francis turbines have fixed pitch runner blades), and can lead to unwanted flow characteristics in the draft tube which can critically weaken performance. Hence, hydraulic losses in the draft tube, and thus the residual swirl exiting the runner, essentially shape the performance curves of the machine (Susan-Resiga et al., 2011).

The primary unwanted flow characteristics in the draft tube, arising from residual swirl, are recirculation of the core and separation of the turbulent boundary layer from the draft tube walls. As discussed, the latter can be somewhat controlled by allowing some residual swirl into the draft tube. However, if there is too much swirl, recirculating flow at the diffuser core will occur. Thus, according to measurements by Clausen et al. (1993), there is a ‘small range of swirl number (the ratio of maximum circumferential to average axial velocity) that avoids both recirculation and separation’ in a diffuser. The interplay between these two flow features, their effects on performance, and the fixed pitch of the runner blades, produce design
constraints on the runner blades.

A high level of residual swirl plays a role in producing another debilitating flow feature in the draft tube called a hub vortex, also known as a vortex rope (Escudier, 1987), or Precessing Vortex Core (PVC). This vortex forms at the core of the flow, along with the recirculation region. It can be stable, and stay near the axis of the turbine, or it can precess, forming a helical shape. The low-frequency periodic precession of the rope can cause pressure fluctuations throughout the machine, noise, vibration, damage, and performance swings (Alligné et al., 2008; Dörfler et al., 2013; Foroutan and Yavuzkurt, Foroutan and Yavuzkurt, 2014; Susan-Resiga et al., 2006, 2010). The stability of the vortex rope depends on the phenomena of vortex breakdown (VB) (Lucca-Negro and O’doherty, 2001). There is no general definition for VB (Susan-Resiga et al., 2006), but Leibovich (1978) describes it as a major structural change of a vortex that occurs due to variations in the characteristic ratio of circumferential to axial velocity. The class of VB, whether that be of the axisymmetric bubble type or the spiral type, will determine if the vortex rope will remain fairly axisymmetric or precess. Studies have been conducted to prevent vortex breakdown in draft tubes by injecting water, and thus momentum, at the hub axis (Bosioc et al., 2010a b; Foroutan and Yavuzkurt, Foroutan and Yavuzkurt, 2012a; Muntean et al., 2008; Susan-Resiga et al., 2010). Susan-Resiga et al. (2010) displayed the elimination of the breakdown in this manner, and hence, the vortex rope was prevented from forming, even though the injection had practically no effect on the overall machine efficiency. Understanding the vortex rope, its breakdown and stability, and methods of preventing it, are all major hydraulic turbine research areas.

Another common, and troublesome, low-frequency vortex structure occurring in hydroturbines, is the inter-blade vortex. Inter-blade vortices form between the runner blades when the flow is not optimally aligned with the leading edge of the blade, such as at off-design conditions (Dörfler et al., 2013). The vortex forms at the inlet to the runner, near the leading edge of each blade (at the shroud or the hub), and convects through the blade channel along the suction side of each blade, and into the draft tube. Inter-blade vortices can cause intense broadband pressure pulsations throughout the machine resulting in strong mechanical vibration in many components including the penstock (Dörfler et al., 2013).

A flow feature which can result in detrimental effects in all multi-stage turboma-
chinery, hydroturbines included, is rotor-stator interaction. In hydroturbines this corresponds to runner blade-guide vane interaction (blade-vane interaction). This phenomenon occurs whenever a runner blade passes behind a guide vane, and thus, passes through the low pressure wake of the vane (Brennen, 2011). This creates an unsteady force on the runner blades. Furthermore, since almost all runners are designed with their number of blades not being an integer multiple of the number of guide vanes, the runner blades do not pass through wakes at the same moment. For instance, the runner is usually designed with a prime number of blades. The non-integer multiple periodicity prevents resonance effects from occurring due to the blade-vane interaction, but at the same time it produces different pressure distributions on each runner blade at any given moment, resulting in asymmetry. In a Francis turbine, the shroud side of the runner blade is much closer to the guide vane than the hub side, and this blade-vane interaction predominately displays itself in that region. In many low-head/high-flow cases, the guide vanes can even extend past the end walls and hang over the leading edge of the runner. Here, the runner-vane interaction may prove to be even more severe.

**Variable Demand Energy Market**

The energy market dictates a variable demand of electrical power due to the intermittent use of electricity by consumers throughout any given day. Also, environmental factors such as weather and season can introduce variation in flow conditions, especially in run-of-river facilities. These variables, along with limited storage capabilities, require greater flexibility of hydroturbines and the ability to produce electricity over a wide range of operating conditions (Alligné et al., 2008; Susan-Resiga et al., 2011, 2010). Thus, operation at off-design conditions is fairly common in the industry. As mentioned, a multitude of debilitating flow features arise at off-design conditions. Many of the flow features discussed can easily cause cavitation to occur if the pressure drops below a critical value (i.e. within the low pressure core of the PVC or interblade vortex). In all of these cases, cavitation will accentuate the problems of performance breakdown, vibration, erosion, damage, and noise caused by these flow features. Furthermore, the low-frequency cavitating flow features can cause severe surges in the machine, on an order which can produce significant electrical power swings (Dörfler et al., 2013). Thus, understanding cavitation is of utmost importance to allow for operation at
the off-design conditions the energy market demands, while mitigating cavitation’s undesirable effects.

1.2 Cavitation

Cavitation refers to the physical process of the inception, transport, and desinence of vapor cavities within an originally homogeneous liquid medium. Cavitation inception occurs in localized regions of the liquid which have fallen below a material property of the liquid called the vapor pressure \( p_v \). At a fixed temperature, if the liquid drops below this very low pressure in some region, a phase change will occur in that region (i.e. vapor cavities will form). This is a similar mechanism to boiling but at roughly constant temperature. The cavities may then be transported through the liquid, interacting with it, and possibly modifying the flow structure if they develop to a relatively large enough scale. In this manner, cavitation within hydraulic machinery can affect performance. When the cavities are transported into a region of pressure higher than \( p_v \), they will collapse, changing phase to liquid once again. The collapse of cavities is usually a violent process which produces high intensity noise, causes severe vibration to nearby solid structures, and can easily pit (microscopic plastic deformation) and erode hard metals such as steel.

The fluid pressure passing the vapor pressure limit does not always imply that inception or desinence of vapor will occur. In some cases, a delay can occur where the pressure can pass \( p_v \) without a phase change. For instance, inertial phenomena can limit inception by not allowing enough time for vapor cavities to become observable (Franc and Michel, 2006). Another factor important to inception are cavitation nuclei. These are microscopic imperfections in the fluid (i.e. dissolved gas, dust particles, etc.) that, through their minute voids in the liquid, act as origin points for cavitation. If a liquid is devoid of such anomalies, then the pressure can become so low prior to inception, that the liquid can support tensile stresses. In most industrial flows the liquid is not so pure, and cavitation occurs due to nuclei far before negative pressures are reached. This dissertation does not focus on the inception problem, but for the interested reader, there is a vast amount of literature on the subject with good starting points being Brennen (2005, 2013); Franc and Michel (2006).

In the present study, the focus is on hydrodynamic cavitation, and in particu-
lar, developed cavitation within hydraulic machinery. Developed hydrodynamic cavitation, which is well established cavitation occurring in flowing liquids, is much less sensitive to inception phenomena such as nuclei content (Brennen, 2013). These cavitation structures are large enough that they interact with, and alter the flow, and occur when localized cavitation events are spatially dense enough to coalesce into large-scale cavitation structures. There are four main forms of large-scale/developed hydrodynamic cavitation. They are:

- **Dense Traveling Bubble Cavitation:** This is a transitional phase between traveling bubble cavitation and large-scale developed cavitation. Here, cavitation events become dense enough that they begin to merge into a large-scale structure downstream of inception, although small individual bubbles still appear at inception and after the collapse of the large-scale cavity (see Figure 1.4).

- **Attached/Sheet Cavitation:** This form commonly occurs in a wake or region of separated flow as a single large-scale vaporous cavity. This is very typical on bluff bodies. It may appear as a smooth transparent surface when separation is laminar and a rough opaque surface as the interfacial layer undergoes transition to turbulent flow (see Figure 1.4).

- **Cloud Cavitation:** This is a structure that arises as the periodic formation, possible shedding, and collapse of cavities which may be dense bubbly mixtures (see Figure 1.4). The periodicity could occur due to shedding vortices or due to periodicity implicit in the flow (such as rotor-stator interaction in turbomachinery). The cloud, which can detach from the surface, may be transported downstream into a region of higher pressure, resulting in coherent and violent collapse. This form of cavitation causes much higher intensity noise, vibration, and damage than other forms of cavitation.

- **Vortex Cavitation:** In localized regions of flow where vortices form, cavitation can occur in the vortex core. This is due to the pressure of the vortex core being substantially lower than the rest of the flow. As more cavitation occurs, the entire vortex can become vapor (see Figure 1.4).

It is of note that cloud cavities often form by shedding periodically from a sheet cavity. The physical mechanism responsible for the shedding is what is known as a
reentrant jet, and it is this mechanism that is thought to be responsible for cavity closure as well. As the interfacial layer of the cavity becomes unstable and turbulent, it turns into a bubbly mixture, which causes some of the flow to recirculate back into the cavity. This recirculation is the reentrant jet, and its periodic impingement upon the cavity, can cause the periodic pinch-off of a cloud cavity which is then convected downstream. The reentrant jet is illustrated in Figure 1.5.

To characterize the extent as to which a flow is cavitating, the cavitation number is introduced. This non-dimensional parameter is the ratio of the pressure head for vaporization to the dynamic pressure of the flow. It can be mathematically defined
in a number of different ways but is commonly defined as

\[ \sigma_{cav} = \frac{p_\infty - p_v}{\frac{1}{2} \rho_l v_\infty^2} \]  

(1.1)

where \( p_\infty \) is the freestream pressure, \( p_v \) the vapor pressure, \( \rho_l \) the density of the liquid, and \( v_\infty \) the freestream velocity. As \( \sigma_{cav} \) decreases the extent of cavitation in the flow will increase.

Another method of describing a non-dimensional cavitation number that is commonly used in the hydroturbine industry is called the Thoma number (\( \sigma \)) which is the ratio of the Net Positive Suction Energy (\( NPSE \)) and the net specific energy (\( E \)) between the inlet and outlet.

\[ \sigma = \frac{NPSE}{E} \]  

(1.2)

The Thoma number is the most common cavitation parameter used in this dissertation. It, and the definition of \( NPSE \) and \( E \) are defined in more detail in Section 2.4. For now, it suffices to know that for a given situation \( \sigma \) is proportional to \( \sigma_{cav} \) (Arndt, 1981), and thus, as \( \sigma \) decreases, the extent of cavitation in the flow increases, just as with \( \sigma_{cav} \).

### 1.2.1 Turbomachinery Cavitation

The investigation of cavitation in liquid turbomachinery goes back as far as 1894, when Charles Parsons experienced, and began studying, cavitation in his steam turbine engine which powered the fastest warship in the world (Foeth, 2008). From this very onset of high speed liquid turbomachinery, cavitation proved to be a problem, causing performance breakdown, vibration, noise, and erosion/damage to components. These detrimental features can occur in all forms of liquid turboma-
Cavitation is not limited to the rotating components of the machine either, as it can form in stationary components as well. It is well known that performance breakdown in turbomachinery is a result of developed cavitation (Arndt, 1981), and all forms of developed cavitation which have been discussed may manifest within turbomachines. Developed cavitation is an inherently unsteady phenomena, and even though performance breakdown occurs under theoretically steady assumptions, it is critical to consider the unsteadiness of cavitation to study noise, vibration, erosion, and damage.

To reduce costs and weight, it is common to design the machine with as small a diameter as possible at a specific power (Brennen, 2011). However, the smaller the diameter, the lower the cavitation number, and this increases the propensity for cavitation. Thus, cavitation is a limiting factor on the design, not allowing designers to reduce the diameter past a point where the deleterious effects of cavitation set in. Moreover, the need to run these machines at off-design conditions results in more cavitation challenges which must be overcome (Avellan, 2004). Furthermore, as turbomachines increase in power and speed, cavitation research becomes more relevant (Arndt, 1981).

**1.2.2 Hydroturbine Cavitation**

Cavitation has presented a problem in hydropower reaction turbines from the onset of hydroelectric generating stations. Originally, the problems of cavitation arose due to poor runner design which allowed sub-vapor pressure regions to occur on the blades, or too much residual swirl to enter the draft tube and allow for cavitation beneath the hub. Sometimes, large amounts of cavitation occurred, even at BEP, and resulted in severe erosion (Figure 1.6), which caused decreased performance over time (Kumar and Saini, 2010), unavoidable repair costs, and significant loss of revenue due to machine down time (Arndt et al., 1989; Bourdon et al., 1999; Gordon, 1992). However, modern turbines usually operate close to cavitation-free at BEP, but will always run into cavitation at even slightly off-design conditions (Figure 1.7). This can severely limit the operation range of the machine. Theoretically, in reaction turbines, cavitation can be avoided completely if the runner is placed a certain distance beneath the tailrace level. However, this distance is usually very deep, and requires large costs for excavation and maintenance (Dixon, 2005).
Consequently, this is never done. Thus, cavitation is unavoidable, especially at off-design conditions, and in this regime it is necessary to understand the effect of cavitation on performance and the unsteady forces it creates (Arndt, 1981). Optimizing efficiency, while minimizing the problems associated with cavitation, are the most important design considerations for modern hydroturbines (Danel, 1959; Klimovich, 1997).

Hydroturbines are following the same trends as other turbomachines, in that increases in power, speed, and time run at off-design conditions, are causing cavitation to become more prevalent (Arndt, 1981; Avellan, 2004). Besides wear
and damage, hydroturbines also possess the other debilitating features of cavitation associated with all turbomachinery, such as noise, vibration, and performance breakdown. In model testing, as the Thoma number is decreased at constant head, and cavitation occurs to a greater extent, the efficiency of the machine reaches a critical point, after which, it drops significantly (Figure 1.8). This breakdown in the performance is due to the development of large-scale cavitation structures that extend up to the runner outlet (Avellan, 2004), and is an important feature of the design and operation of the machine. Thus, cavitation is critical for understanding the limits of machine operation.

The cavitation structures that occur in a Francis turbine usually occur when the previously discussed flow features drop the pressure below the vapor pressure. Cavitation of these flow structures commonly results in more severe pressure oscillations than non-cavitating flow. The most common areas for cavitation on modern Francis turbines are on the leading edge of the runner, the trailing edge suction-side of the runner, the interblade vortex, the hub vortex, and the gaps and overhang of the wicket gates. The hub vortex core can cavitate and precess (Figure 1.9). As Thoma number is lowered, the precession begins to subside and a large axi-symmetric torch-like cavitation structure is formed (Figure 1.9) which can pulsate. Draft tube vortex cavitation is known to reduce the natural frequency of the entire system which can cause large pressure surges, and thus damage and electric power swings, when the vapor collapses (Dörfler et al., 2013). The

Figure 1.8: Efficiency ($\eta$) breakdown of a Francis turbine. The net specific energy and guide vane opening angle are held constant while the Thoma number is reduced. (Avellan, 2004).
Precessing vortex core occurs at \( \sigma = 1.24 \) and the torch occurs at \( \sigma = 0.52 \).

Interblade vortex is also known to cavitate and cause damage to the shroud (Figure 1.10). Cavitation on the runner blade is the most likely candidate for causing performance breakdown. Cavitation beneath the overhang of the guide vane can affect the blade-vane interaction pressure fluctuations and can even impinge on the leading-edge of the runner, causing damage.

Reduced scale model tests present a unique challenge to cavitation studies in hydroturbines. Since model tests are much smaller, there is a lack of Froude similarity between the model and the full-scale prototype (Avellan, 2004). Essentially, this means that the hydrostatic pressure gradient, due to gravity, can not be scaled. Hence, gravity may cause different vapor structures, and even performance effects, in the prototype than what is seen in the model. For this reason, model performance guarantees of cavitation tests are only attempted to be scaled up for values of Thoma number greater than that which would cause a 0.5 percent
drop in efficiency (IEC, 1999). Otherwise, because the scaling laws for very large vapor structures are unknown due to lack of Froude similitude, the performance predictions would be incorrect. This remains a problem in the industry because the full-scale prototypes are usually closed off to any type of visual observation, and it is not always clear what type of cavitation is occurring. Attempts have been made to develop acoustic detection of cavitation in hydroturbines for these reasons (among many others) (Escaler et al. (2006), Escaler et al. (2006)). Multiphase Computational Fluid Dynamics may prove to be beneficial to understanding differences in cavitation between model and full-scale turbines, due to lack of Froude similarity, because it is not a problem to run the simulations at both scales and observe the differences.

1.3 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the numerical solution of the fundamental equations of fluid mechanics applied to a wide range of scientific and engineering problems. In particular, it has been applied extensively to turbomachinery since its inception (Denton and Dawes, 1998). It has been used for the analysis of propellers, pumps, and turbines, and has become a critical component of the turbomachinery design process. CFD is usually more economically attractive than experiments, and often can provide more complete data of the flow field. However, turbomachinery CFD still has limitations, which are critically important to understand, and which are given in detail by Denton (2010). In short, to make CFD a viable tool for turbomachinery design, it is extremely important to consider the potential errors of CFD through processes such as code verification, and validation of results with experiments. Thus, CFD is a comparative tool, and not a stand-alone resource for predicting performance (Denton, 2010).

Modern turbomachinery CFD can predict steady and unsteady flow through the three dimensional stages of a device. Steady flow is usually computed through periodic single-blade passages of each stage, with acceleration terms added to the equations of motion to account for blade rotation speed. Only computing a single blade of each stage, and assuming the flow similar in every other blade passage of that stage, reduces the computational cost a great deal and commonly provides quality performance predictions (Denton and Dawes, 1998). However, due to non-
multiple periodicity between the stages, as well as unrealistic wakes forming because
the blades are not actually rotating past one another, a circumferential average
must be taken at the interface between the stages. This is commonly done with a
mixing plane (Denton and Dawes, 1998), which is a circumferential average of the
flow variables, such as pressure, and tangential/radial velocities, at the interface
between the stages. Unsteady simulations are most commonly conducted using the
full machine geometry, including all blades of each stage, with meshes that rotate
relative to one another.

Turbulence modeling is usually an important consideration to CFD because
solving the complete Navier-Stokes (N-S) equations and resolving all scales is much
too computationally expensive for practical flows at this point in time. Therefore,
the Reynolds-Averaged Navier-Stokes (RANS) equations are solved and turbulence
models are necessary to close the system of equations (Wilcox, 1998). The RANS
models are also used to simulate unsteady flow to a moderate degree of accuracy
in what is called Unsteady RANS (URANS) (Ferziger and Perić, 1996). Another
method is to use the filtered Navier-Stokes equations to capture the large-scale
eddies and model the small-scale structures (Pletcher et al., 2012). This is called
a Large-Eddy Simulation (LES) and requires more computational resources than
URANS but provides more realistic results. Finally, a Detached-Eddy Simulation
(DES) is a hybrid method that uses URANS near the walls where the mesh is fine
and LES away from the walls to capture the large-eddy structures accurately. DES
has become very popular with practical turbomachinery computations.

Single-phase CFD has been used in the hydraulic turbine field since 1978 (Keck
and Sick, 2008; Sick et al., 2009) and has become an important design tool for the
industry. Numerical hill charts (efficiency curves across different design points) for
entire turbines have been computed at over 200 operating points and match well
with experiments (Drtina and Sallaberger, 1999). At this point many of the single-
phase flow features discussed, such as the vortex rope, vane-runner interaction,
vortex shedding, and inter-blade vorticies have been simulated in fully-coupled
3D URANS calculations (Keck and Sick, 2008). CFD in hydroturbines is now
beginning to use DES more frequently to compute these single-phase flow features
(Foroutan and Yavuzkurt, 2012a b, 2014). Currently, one of the most active areas
of hydroturbine CFD research is concerned with the computation of cavitating
flows (Keck and Sick, 2008).
1.3.1 CFD Applied to Cavitating Flow

One of the most common approaches to numerical prediction of multi-species flow is the locally homogeneous mixture formulation. In the case of cavitating flow, this is a homogeneous mixture of vapor and liquid. Variables, such as the density, are treated as a weighted average of vapor and liquid at any given cell in computational domain. This method requires as many continuity equations as there are species in the flow, to model the separate mechanics of each constituent, and to capture the correct local equilibrium mixture properties. Furthermore, mass transfer models are inserted as source terms in the continuity equations to govern the phase change between the constituents. A momentum equation may be used for each species as well, but this would require modeling of the interfacial dynamics between species (Kunz et al., 2000). However, assuming no-slip and a common thermodynamic pressure between species residing in the same control volume, allows for a unified velocity field, and thus a single momentum for the entire mixture, without further interfacial modeling (Kunz et al., 2000). These assumptions can be justified for large-scale cavitation because, here, surface tension is negligible and the species are well separated (Kinzel, 2008). The Eulerian homogeneous mixture approach, governed by a single momentum equation of the mixture, along with a continuity equation for each constituent, and models to govern mass transfer between species, has become one of the most popular and reliable methods to simulate large-scale cavitating flows (Kunz et al., 2000; Merkle et al., 1998; Venkateswaran et al., 2002).

Due to the extra equations, multiphase flow computations require more computational resources than single-phase flow. Furthermore, it is common knowledge (Brennen, 2013; Venkateswaran et al., 2002) that for a liquid-vapor mixture, the local speed of sound can be reduced to very low values (Figure 1.11). Consequently, even for low-speed flows, compressibility can be an important factor to predicting damage (Brophy et al., 1985) and flow dynamics (Venkateswaran et al., 2002). Therefore, another hurdle is that computational methods for multi-species flow must be able to operate across all ranges of Mach number. However, through advancements in computer technology, research efforts (Kunz et al., 2000; Merkle et al., 1998), and the development of preconditioning methods to compute low-speed compressible flows with density-based flow solvers (Venkateswaran et al., 2002, Venkateswaran and Merkle, 1999), multiphase flow solvers are now dependable.
Figure 1.11: Mixture sound speed versus vapor volume fraction for equilibrium saturated steam at 300 K (Venkateswaran et al., 2002).

for computing large-scale cavitating flow in practical geometries, across all Mach numbers.

Soon after this technology became available, cavitation simulations were conducted in turbomachines, such as pumps (Medvitz et al., 2002) and propellers (Lindau et al., 2005, 2009), using RANS turbulence modeling. Cavitating flow investigations were then extended to URANS and DES (Kinzel et al., 2007; Kunz et al., 2003) and various multiphase flow solvers were compared for propulsor flow (Salvatore et al., 2009). Other studies were conducted to compare the mass transfer cavitation model’s effect on the solution (Morgut and Nobile, 2011; Morgut et al., 2011; Park et al., 2009).

Based on success with CFD modeling of other cavitating turbomachinery flow (Hosangadi et al., 2004; Lindau et al., 2005, 2012; Medvitz et al., 2002), it is fair to say that tools currently in use are capable of cavitating flow CFD analysis through hydroturbines. It is then straightforward to use results of this analysis to obtain accurate installed performance predictions, as well as indications of potential cavitation damage and how it might be avoided. For the scales of vapor cavity volume and duration that are needed to affect hydroturbine performance, there are no physical limits on cavitation imposed by nucleation. Therefore, this type of cavitation is amenable to being modeled with the homogeneous mixture model, using a single momentum equation as proposed by Kunz et al. (2000).
Hydroturbines

Although there is a great deal of CFD research for hydroturbines, and a wealth of CFD analysis of cavitating flow in turbomachinery, investigations of hydroturbines utilizing multiphase computational fluid dynamics are rather scarce. This is even more surprising when we recall how large of an impact cavitation makes on the hydroturbine industry. The most common approach to analyzing cavitating flows in the past (and these methods are still predominant in the present) was to compute the single-phase problem, and then examine the pressure field for pressures which were below the vapor pressure (Alligné et al., 2009; Keck and Sick, 2008; Rudolf, 2009; Rudolf et al., 2012). Now, this approach of locating the pressure zones below vapor pressure and then adjusting the design of the machine to reduce or eliminate them entirely, has proven to be extremely beneficial to the industry. The cavitation-free operating range has been widened in modern turbines (Dörfler et al., 2013). However, the improvements have already been taken up by operators pushing the limits of the turbines (Dörfler et al., 2013). When cavitation does occur, single-phase computations have no ability to predict the performance breakdown of the machine. Furthermore, the shape and size of the cavity is not equivalent to the corresponding single-phase sub-vapor pressure region, as the large scale cavities, being a much lower density than the water, are deformed and transported in the flow. A fluid particle’s path can be drastically altered due to the presence of vapor, whereas the single-phase computation will not capture this. Moreover, the unsteady effects amplified by (and occurring as a result of) cavitation, such as vibration, noise, and damage, have no chance to be analyzed with single-phase CFD. Here, the published research concerning cavitating CFD in hydroturbines will be reviewed in detail. All of these researchers applied the locally homogeneous mixture approach to the Navier-Stokes equations, along with various forms of mass transfer models to govern production and destruction of vapor.

One of the first attempts to employ multiphase CFD to calculate the cavitating flow field within a hydroturbine was by Susan-Resiga et al. (2003). A periodic, steady computation of a Francis runner blade was studied with the cavitation model proposed by Schnerr and Sauer (2001). Only one Thoma number was investigated, and this was for an on-design condition which did not have enough cavitation on the runner to affect the performance. Experimental results even displayed a slight increase in torque at that Thoma number, due to cavitation on the suction-side of
the blade. The simulation successfully reproduced this torque increase, and also
the cavitation displayed qualitative similarity to experimental photos. Additionally,
the pressure distribution on the cavitating blade was noticeably different when
compared to the single-phase results.

At just around the same time as Susan-Resiga et al. (2003), multiphase CFD
was used for the first time to predict the performance breakdown in a Francis
runner single-blade passage (Lipej et al., 2002). However, these results were not
quantitatively compared to experiments. Qualitative cavitation structures compared
well with the experimental images for both the Francis turbine blade and a bulb
turbine runner blade. Similar results were shown again by Jošt and Lipej (2011)
for the bulb turbine.

Panov et al. (2012) simulated steady cavitating flow in two different model-scale
Francis hydroturbines with 4 different cavitation models. To reduce the computa-
tional cost, the computational mesh was generated to have single, periodically
bounded, blade passages for the guide vanes and runner blades. The draft tube
was mentioned to be simulated as well, but it is not clear whether this was the full
draft tube with an elbow, a modified axi-symmetric/periodic slice, or something
else. Either way, the geometry and flow field of the guide vane and draft tube, and
cavitation in these regions (if there was any), was not displayed or discussed. The
3 stages simulated were connected by mixing planes. An inlet/outlet boundary
condition method was proposed which allowed the specific energy to be specified,
rather than the traditional CFD approach of fixed flow rate inlet and fixed pressure
outlet conditions. It was mentioned that this approach is better suited for cavitating
hydroturbines because the head is usually constant while the flow rate can vary
sharply as cavitation increases. A density-based, isothermal, compressible solver
with preconditioning was used to compute the flow. The performance results com-
pared well with the experimental data for both turbines, and cavitation breakdown
of performance was displayed, although was predicted to occur at higher Thoma
number than in the experiments. The two meshes used for each case were both very
course, and it could be questioned as to whether definitive mesh convergence was
actually proven. Iso-surfaces of volume fraction in the runner domain, highlighted
cavitation on the aft suction-side of the blade, which qualitatively agreed with the
model testing photos. All four cavitation models tested showed little variation
in the performance results, and it was concluded that the choice of model was
Figure 1.12: Efficiency cavitation breakdown curves for experiment and 4 different CFD cavitation models (Panov et al., 2012).

insignificant to that steady study (see Figure 1.12).

Stein et al. (2006) presented single and cavitating simulations of the guide vanes, runner, and draft tube of a Francis turbine at part-load. The results displayed the difference of the draft tube pressure pulsations between single-phase and cavitating flow. An extremely dense mesh of 35 million nodes was employed and resulted in performance data which compared favorably with the experiments. Furthermore, the results were compared with extensive Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV), and wall pressure measurements, with good agreement. The focus of the cavitating flow analysis was in the draft tube.

Dörfler et al. (2010) computed two-dimensional, axisymmetric cavitating flow in a draft tube to investigate full-load surge in Francis turbines. The 2D computation was justified by the circumferential symmetry of the full-load cavitating vortex usually observed in experiments. The cavitating results produced a much larger region of vapor in the hub vortex than the single-phase results suggested (due to the vapor transport), and also displayed a standing wave on the free surface which can not be predicted by single-phase flow. The traditional 1D hydroacoustic analysis of the mass flow gain in the draft tube, which is mostly agreed to be the mechanism producing large pressure pulsations (surge) in cavitating Francis turbines at high flow rate, assumes that the mass flow gain is controlled by the downstream flow rate. The authors claim, and display through their results, that the conventional 1D models do not coincide with their simulations, and a modified stability analysis
is necessary. Furthermore, the results display that the mass flow gain is critically dependent to the upstream flow rate (at the runner exit), as long as a delay time is considered, rather than the downstream flow rate.

After Dörfler et al. (2010), other attempts were made to utilize CFD to study surge (pressure/flow rate pulsations due oscillations of cavity volume in the draft tube), by extending the 1D hydroacoustic model to be coupled with 3D URANS multiphase CFD (Chirkov et al., 2012, 2014; Mössinger et al., 2014; Panov et al., 2014). The approaches taken by these studies are referred to as hybrid 1D-3D CFD models. Each of these investigations employs unsteady cavitating CFD simulations through the guide vanes, runner, and draft tube (stay vanes as well in the case of Mössinger et al. (2014)), and uses the 1D hydroacoustic model to compute the fluctuations through other components such as the penstock, spiral casing, and tailwater. The 1D and 3D approaches are coupled through boundary conditions. A major benefit of using a hybrid method is that, for the simple 1D approach, terms such as ‘cavitation compliance’ and ‘mass flow gain factor’ must be determined with experiments or a series of steady cavitating CFD simulations at various operating points. Using multiphase CFD to directly simulate the cavitation regions, allows for detailed information pertaining to the unsteady cavity dynamics in the draft tube, and thus, the previously mentioned 1D model terms are unnecessary (the source of pulsations is captured). It should be noted that all of these studies only used periodic single-blade passages for CFD of the vanes and runner, but used the full 3D draft tube.

Panov et al. (2014) analyzed model and prototype scales (although for different runner geometries between the scales, the prototype was not boundary layer resolved, and no direct comparison was attempted between the scales), and displayed agreement with experiments. For the model case, draft tube cavity volume was shown to vary at a frequency corresponding to pressure and flow rate fluctuations in the machine (see Figure 1.13). For the prototype case, the fluctuations of cavity volume, due to their lack of periodicity, did not seem to correspond with the pressure oscillations. The influences of Thoma number, flow rate, and penstock length, on the results, were also displayed. Mössinger et al. (2014) conducted a similar analysis to Panov et al. (2014), but for an overload (rather than full-load) prototype turbine that also included spiral casing/tailwater effects in the 1D model. Periodic oscillations in cavity volume and pressure were obtained.
A number of investigations on multiphase CFD of a particular Francis turbine, ranging from steady to unsteady results, have been conducted (Liu et al., 2009, 2005; Wu et al., 2011). The spiral casing, all of the stay vanes, guide vanes, runner blades, and the entire draft tube were considered for the steady cases. All of these simulations were run with the original ‘full cavitation model’ of Singhal et al. (2002) or a slightly modified version of this model, the RNG $k$-$\varepsilon$ turbulence model with either RANS or URANS, and a pressure-based solver. The initial study presented steady performance results of the full-scale prototype which captured the performance breakdown due to cavitation (Liu et al., 2005). It was stated that no cavitation occurred in any regions besides the runner and draft tube, and that the cavitating regions compared well with the experiments. However, no experimental data was provided or directly compared in this preliminary study. Expanding on that study, Liu et al. (2009) conducted steady computations of the model-scale machine and compared them with experimental results. A mesh refinement study was done to display that the torque was not modified when the mesh was further refined. Part-load, BEP, and full-load conditions were all computed, and the performance breakdown was captured. The simulation compared well with experiments and the critical Thoma number was predicted close to the empirical data. Cavitation structures of the hub vortex and on the runner blades compared qualitatively well to experimental photos. Some preliminary unsteady results were shown, but these were presented in more detail in the next paper from this group (Wu et al., 2011).

Wu et al. (2011) provided cavitating URANS computations of the guide vanes,
runner, and draft tube of a Francis turbine and compared the amplitude and dominant frequencies of the draft tube pressure fluctuations (both single-phase and cavitating) to the experiments. These simulations were run for both part-load and a full-load cases. Both the single-phase and the cavitating results were compared to the empirical data at two specified locations on the draft tube walls. As would be expected, the location closer to the runner exit always displayed larger amplitude fluctuations than further downstream in the draft tube, but both displayed the same frequency. The cavitating results showed both lower amplitude and dominant frequency than the single-phase results. The results also showed that the fluctuations decreased in amplitude but increased in dominant frequency as the flow rate increased, although not nearly to the degree in which the experiments did so. Finally, the qualitative features of the hub vortex cavitation, precession and torch-like, compared well with the experiments but with some noticeable length and diameter differences. An example of the draft tube pressure fluctuations, and an image of the vapor on the runner and in the draft tube, are shown in Figure 1.14.

Wang and Chang (2010) computed cavitating flow in periodic passage of a coupled Francis runner blade and guide vane. The simulations were conducted at BEP and vapor was displayed on the suction-side of the runner blade. The torque breakdown was captured although it occurred at higher Thoma number than the experiments. The main purpose of this paper was to present the notion that water
quality will affect the cavitation in hydroturbines, by modifying the nuclei content. It was shown that more nuclei are present in lower quality water, and that this would make turbines more susceptible to cavitation. This was displayed in the numerical results by varying the vapor pressure and displaying more cavitation on the runner blade. While water quality may be important to determining the cavitation characteristics of a turbine, this notion is not considered any further in this dissertation.

An LES study of a cavitating Francis turbine, Guo et al. (2007) compared single and two-phase results to experimental draft tube measurements. For the single-phase flow, both the axial and tangential velocities were compared with the experiments, however, the empirical data points were not available in the most critical region, beneath the hub, and were only closer to the walls. The cavitating results were obtained for two cavitation numbers and were only qualitatively compared to the experimental photos beneath the hub, to which they displayed good agreement. Also, the pressure pulsations beneath the hub were shown to be lower in amplitude for smaller cavitation number.

Jošt and Lipej (2009) presented cavitating results for the runner and the draft tube of a Francis turbine using both URANS and Reynolds Stress turbulence models. The results were qualitatively compared with experimental photos and it was mentioned that the rope was straight for both cases, and not precessing, as it was in the experiments. The pressure pulsation amplitudes were decreased relative to the single-phase (as expected from the previously mentioned studies), and the frequency did not change much. Furthermore, the pressure pulsations of the Reynolds Stress model calculation were said to be irregular, possibly due to the coarseness of the mesh, and it was mentioned that further investigation into this matter was necessary. Jošt and Lipej (2011) extended this work to a LES where only the cavitation in the draft tube component was computed with an unsteady inlet. The cavitating LES gave draft tube pressure pulsation amplitudes that were much smaller than the experiments.

A numerical simulation was conducted for cavitating flow in a part-load Francis turbine (Zhang and Zhang, 2012) using the Kunz mass transfer model (Kunz et al., 2000) and URANS turbulence modeling. The entire machine was computed. Contour plots of pressure and volume fraction were shown but these results were not compared with experiments. The purpose of the research was to demonstrate
that the software had the capability to compute multiphase flow in an entire hydroturbine.

Multiphase flow in a Kaplan turbine has also been predicted by the simulations of Nennemann and Vu (2007). The guide vane-runner blade interaction was studied to predict and understand the periodic formation and collapse of cavitation structures in the turbine. Results of volume fraction shown on the blades compared well with photos of the experiments, and unsteady pressure measurements were consistent with the CFD results. The CFD results predicted a low pressure dip due to the tip gap vortex which the experiments did not measure. This was said to either be an error in the positioning of the sensor or the fact that the CFD was for the prototype-scale while the experiment was model-scale. This led to further questions as to how the scaling affects features such as cavitation intensity, vortex intensity, and pressure fluctuations.

Zuo et al. (2014) conducted the first detailed stability analysis, using 3D multiphase CFD, of the cavitating draft tube vortex in a Francis turbine at different operating conditions. URANS simulations from the guide vanes through the draft tube exit were conducted and pressure fluctuations were monitored in both the guide vane and draft tube regions. The amplitudes and frequencies of the oscillations agreed favorably with experiments. A Batchelor instability analysis was utilized to display the instability zones for each design point simulated, which showed different stability characteristics between the cases. Images depicting the varying vortex structure amongst the operating conditions were also provided.

Partial and overload cavitating URANS simulations were presented by Tamura et al. (2014) and Shingai et al. (2014), for the guide vanes through the draft tube of a model Francis turbine. The focus of the study was on the draft tube cavitation and its affect on the flow, especially in the overload condition. Both the partial and overload pressure pulsations in the draft tube displayed fairly good agreement with model experiments. The partial-load case captured the helical cavitating vortex and its rotation rate when compared with the experiments. For the overload case, the experiments displayed a large bell-shaped torch cavity underneath the hub nose cone. A very long period draft tube pressure oscillation was observed as well, that occurred over 15 runner revolutions. Initially, the torch shape began to extend far into the draft tube until it approached the draft tube elbow. At that point, a recirculation region formed in the core of the draft tube (thought to be due to
the cavitation which brought about flow separation at the draft tube elbow, that started to collapse the hub cavitation. The cavity collapsed enough to almost form a helical structure. Flow then reattached at the draft tube elbow, and the cavity returned to the long bell-shaped torch structure once again. The CFD simulations captured this phenomena well, and comparisons between experiment and CFD images, as well as draft tube pressure readings, were provided.

As far as utilizing multiphase CFD to improve the cavitation characteristics of a hydroturbine, only the study of Wei et al. (2014) exists in the published literature. A periodic URANS calculation through the guide vanes and runner of a Francis turbine, displayed cavitation on the runner blade. The simulation was analyzed for the lowest pressure on the runner blade, and a perforation was placed at that location. The simulations were run again, and showed a large reduction in vapor on the blade surface. Moreover, similar experimental model tests were conducted, with and without the perforated blade. The perforation location was chosen with the CFD results. The experiments displayed similar reduction of vapor on the blades and also changed the shape and cloudiness of the draft tube hub vortex. Efficiency and critical Thoma number were unchanged when the perforation was added in the experiments, but the noise was greatly reduced, and pressure pulsations in the draft tube were slightly reduced. Figure 1.15 displays the reduction in cavitation on the runner blade when the perforation was added to the CFD simulation.

Some other studies, which are not directly of hydroturbine components or are only single-phase flow, but may still be applicable, will now be concisely summarized. Cavitation was computed in a conical diffuser and compared to experiments in
an attempt to understand cavitation surge effects in a draft tube by way of a one-dimensional stability model (Chen et al., 2010). A similar model was used by Alligné et al. (2009) for a hydroturbine calculation, however, the CFD used for the input to the stability model of Chen et al. (2010) was single-phase. For instance, the hub vortex volume variation, was calculated by a region bounded by the vapor pressure instead of a percentage of vapor volume fraction. A number of computational and experimental studies in a diffuser and converging-diverging nozzle were conducted by Rudolf (2009); Rudolf et al. (2012), with relevance to the cavitating draft tube vortex. While those two studies only computed single-phase flow, Rudolf et al. (2012) presented results from a multiphase simulation.

1.4 Scope of Contributions

It is clear from the preceding literature review that, although a number of studies have been conducted, the computation of cavitating flow in hydroturbines is still in its infancy. Cavitation is a local phenomenon which can drastically alter the performance of the hydroturbine, even when vapor only makes up a small fraction of the total fluid within the machine. Thus, to accurately capture the details of performance breakdown it is necessary to thoroughly resolve the localized regions of vapor relative to the large scales of a hydroturbine. Therefore, while the computational mechanics foundation to accomplish these simulations is already available, the process of accurately calculating complex multiphase flow fields in multi-stage hydraulic machinery is still a formidable challenge. Moreover, the lack of Reynolds and Froude scaling in the reduced-scale model tests has always presented a problem to the industry. Standards have been developed for scaling the single-phase results to the full-scale machine, but they are only acceptable for small amounts of cavitation which do not greatly affect the performance. In addition, forecasting what the cavitation structures will look like at full-scale, from the model testing, is difficult. However, running at both scales, and comparing the results to gain insight into these problems, is something that CFD can accomplish with relative ease. Yet, there is a gap in the cavitating hydroturbine CFD literature, as there have been no direct comparisons between model and full-scale cavitating CFD results.

For single-phase flow, detailed CFD analyses of the stage-by-stage performance
decomposition of hydroturbines have been published (Jošt and Lipej, 2009; Vu and Retieb, 2002). These studies have provided important information pertaining to the major loss mechanisms in modern hydraulic turbines. In contrast, a similar study has yet to be conducted with cavitating CFD. Questions arise as to what the key loss mechanisms are as cavitation breakdown occurs.

A number of unsteady cavitating CFD studies have been conducted on hydroturbines and published in the literature. Only two of these studies used an LES/DES approach in an attempt to accurately capture the dynamics of the cavitating vortex rope. In these two cases, no attempt was made to simulate more than the draft tube with a cavitating flow LES. Furthermore, all of the unsteady cavitating CFD studies primarily focused on low-frequency flow phenomena in the draft tube. No data is published for the runner torque fluctuations due to cavitation on the blades nor is the maximum pressure on the runner blades analyzed in a cavitating CFD case. Simulating, analyzing, and understanding cavitating flow in the runner that may lead to blade erosion and wear, can ultimately be one of the most promising applications of large-scale cavitation CFD to the hydroturbine community. However, to accurately obtain data for the low-frequency draft tube phenomena, a large number of runner revolutions (possibly 10-100 revolutions) is required, and this usually amounts to taking larger time-steps due to computational resource limits. Perhaps the choice of many researchers to analyze the draft tube flow in detail has led to insufficient time-step size, insufficient runner mesh spacing, or inadequate simulation methods (in the case of many studies which only compute periodic vane and runner regions) for the resolution of the potentially damaging forces induced by cavitation collapse on the runner blades. As vapor collapse on the runner blades is an extremely localized event occurring over very short time intervals, and can be highly dependent on factors such as blade-vane interaction, it is possible that previous studies did not attempt to capture these events anyhow.

In this dissertation the cavitating flow field within a model Francis hydroturbine is computed with high-fidelity multiphase CFD simulations and compared with experimental reduced-scale model results. In addition, simulations of the full-scale cavitating turbine will also be conducted and compared with the performance results from the experiments that have been scaled up by the industry standard. Steady-periodic methods are used to obtain the cavitation breakdown results for the steady-state performance. The computations are extended to lower Thoma
numbers than the experiments to uncover the differences in cavitation performance breakdown between model and prototype scales. The steady-periodic results are also used to conduct a stage-by-stage performance decomposition, allowing for the analysis of each individual component’s contribution to the losses in the machine. Next, to obtain unsteady cavitating results for the entire machine, a DES is employed. The differences between single-phase flow, cavitating flow, model, and prototype scales, are thoroughly investigated in both the time and frequency domains. Pressure is analyzed in both the draft tube and the penstock. The runner torque and the maximum pressures on the runner blade surfaces are also investigated. Commentary is then given on potential erosion and wear to the hydroturbine blades.

1.4.1 Layout of Dissertation

The remaining chapters of this dissertation proceed as follows. Chapter 2 provides the methodology and approach to computing steady-periodic cavitating flow in a hydroturbine. The computational approach to cavitating flow as well as the methods employed to simulate multi-stage hydroturbines with periodic methods are outlined. Next, the test case is described, including the computational mesh and boundary conditions. Then, the experiments to which the simulations will be compared to, are explained. Chapter 3 presents performance results of the cavitating steady-periodic calculations and the stage-by-stage performance decomposition. Results are presented for the model and prototype scales and discussion is provided with the results. Chapter 4 gives the methods behind the unsteady computations, describing segregated flow solvers, DES, the geometry and mesh of the entire machine, and methods of measuring unsteady flow in hydraulic turbines. Chapter 5 presents the unsteady results in the time and frequency domains. Torque and pressure fluctuations are analyzed and discussion is provided on potential cavitation wear and erosion. Finally, Chapter 6 concludes the dissertation with a summary, presentation of the unique contributions and original findings, and an outline of future work in this area.
2

Steady-Periodic Methods and Approach

2.1 Overview

The primary objective of this dissertation is to compute and analyze cavitating flow in both the reduced-scale model and full-scale prototype of a Francis hydroturbine. The computational results are compared to experimental data, and differences between the model and prototype results are discussed. Steady-periodic results are expected to provide the accurate steady-state performance behavior of the machine as cavitation breakdown occurs. This chapter describes, in detail, the steady-periodic methods employed to achieve these goals. First, the computational approach to the steady-periodic homogeneous mixture simulations, with mass-transfer modeling, is presented. Then the solution strategy is discussed. Next, the solution domain and the boundary conditions for the calculations are introduced. Finally, the experiments to which the computations are compared to, are described along with a discussion on the industry standards for scaling performance specifications between model and prototype scale hydroturbines.

2.1.1 Equations of Motion for a Mixture

The mathematical theory for the continuum mechanics of non-reacting mixtures is a well-established approach for developing the equations of motion for a multi-constituent continuum (Romano and Marasco, 2010). Forgoing the derivation, the general continuity and momentum equations for a mixture of an arbitrary number of fluid constituents are

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  

(2.1)
\[ \frac{\partial \rho v}{\partial t} + \nabla \cdot (\rho v \otimes v) = \nabla \cdot \mathbf{T} + \rho \mathbf{b} \]  

(2.2)

where \( \rho \) is the density of the mixture, \( v \) is the mixture velocity, \( \mathbf{T} \) is the Cauchy stress tensor, and \( \mathbf{b} \) is a body force per unit mass. These equations completely describe the local mechanics of the hypothetical fluid mixture.

### 2.2 Computational Approach to Cavitating Flow

While Equations (2.1) and (2.2) completely describe the mechanics of the mixture, they can not be numerically solved unless they assume some a priori knowledge of what the mixture variables, such as \( \rho \), are at some point. Consequently, for the computation of multiphase flow, Equations (2.1) and (2.2) are commonly split into separate continuity and momentum equations for each constituent. However, in this approach, to numerically solve the momentum equations for each species, a priori knowledge of the inter-facial forces between each fluid must be known, and thus must be modeled (Grogger and Alajbegovic, 1998). Therefore, to eliminate the need for these models, a further assumption is commonly made about the behavior of the fluid-fluid interfaces. If it is assumed that the interface between species is always in thermal and dynamic equilibrium (meaning a unified velocity field exists), only one momentum equation is required (Kunz et al., 2000). This is called the homogeneous mixture model (Toumi et al., 1999) and the justification for this assumption was laid out in Section 1.3.1.

Density-based (i.e. coupled) CFD algorithms were originally developed for compressible flows. However, these algorithms were eventually extended to operate across all Mach numbers through the method of artificial compressibility, which is also known as preconditioning (Chorin, 1967; Turkel, 1999). Preconditioning is a technique that uses so-called pseudo-time derivatives in the equations of motion, which maintain well-conditioned eigenvalues of the system across all Mach numbers. It does so by pre-multiplying these pseudo-time derivatives by a preconditioning matrix \( \Gamma_p \). The pre-multiplication modifies the eigenvalues of the system to ensure that the acoustic wave speeds of the fluid are on the same order as the convective speed. When the solution reaches convergence at each physical time step, the pseudo-time derivative terms should not have any effect on the solution.

Due to the wide-range of Mach numbers experienced in liquid-vapor mixtures,
preconditioning is a vital technique to maintain efficiency and accuracy across all flow regimes. Preconditioning was extended to multiphase flow algorithms by Merkle et al. (1998), Kunz et al. (2000), and Ahuja et al. (2001). By analyzing the eigenvalues of preconditioned two-phase systems with a perturbation method, Venkateswaran et al. (2002) showed that, although these preconditioning formulations were unique, they all maintain efficiency and accuracy across all flow regimes, as with the single-phase approaches.

The vector equation governing unsteady RANS isothermal compressible flow of a liquid-vapor locally homogeneous mixture, with mass transfer between the two phases, is

$$\frac{\partial \mathbf{Q}_c}{\partial t} + \Gamma^p \frac{\partial \mathbf{Q}}{\partial \tau} + \nabla \cdot \mathbf{E} = \mathbf{H}$$  \hspace{1cm} (2.3)

where the physical time derivative, flux vectors, and source terms are defined with

$$\mathbf{Q}_c = \begin{bmatrix} \rho_v \alpha_v \\ \rho_l \alpha_l \\ \rho v_i \end{bmatrix}, \mathbf{E} = \begin{bmatrix} \rho_v \alpha_v v_j \\ \rho_l \alpha_l v_j \\ \rho \alpha_v v_i + p \delta_{ij} - \tau_{ij} \end{bmatrix}, \mathbf{H} = \begin{bmatrix} -(\dot{m}^+ + \dot{m}^-) \\ \rho g_i \end{bmatrix}.$$  \hspace{1cm} (2.4)

Here, \(v_i, p, \) and \(\rho\), are the velocity components, pressure, and the density, respectively, of the mixture. The variables \(\alpha_v\) and \(\alpha_l\) are the vapor and liquid volume fractions, respectively, given as

$$\alpha_v = \frac{V_v}{V}, \quad \alpha_l = \frac{V_l}{V}, \quad \alpha_v + \alpha_l = 1$$  \hspace{1cm} (2.5)

where \(V_v\) is the local volume occupied by vapor, \(V_l\) is the local volume occupied by liquid, and \(V\) is the local volume. The mean viscous stress tensor \(\tau_{ij}\) is defined as

$$\tau_{ij} = (\mu + \mu_T) \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \delta_{ij} \frac{2}{3} \frac{\partial v_k}{\partial x_k} \right)$$  \hspace{1cm} (2.6)

where \(\mu\) and \(\mu_T\) are the dynamic molecular viscosity and the turbulent eddy viscosity. The source vector \(\mathbf{H}\) is composed of the mass transfer rates (per unit volume) \(\dot{m}^+\) and \(\dot{m}^-\), and the components of the gravitational acceleration \(g_i\). While only a single velocity field, and single pressure field, are required for the locally homogeneous mixture model, the mixture density and molecular viscosities are defined in terms
of their corresponding vapor and liquid variables $\rho_v, \mu_v, \rho_l, \mu_l$, by the following weighted-average equations:

$$\rho = \alpha_l \rho_l + \alpha_v \rho_v$$

$$\mu = \alpha_l \mu_l + \alpha_v \mu_v$$  \hspace{1cm} (2.7)

The evaluation of $\mu_T$ is based upon the turbulence model and is discussed in Section 2.2.3. Further, isothermal phasic equations of state $\rho_v = \rho_v(p)$ and $\rho_l = \rho_l(p)$ are necessary to close the system of equations.

Equation (2.4) represents physical terms of the phasic continuity equations with mass transfer (rows 1 and 2), and the physical terms of the single-momentum equation of the mixture (row 3). The variables that make up the unphysical pseudo-time derivative term in (2.3) are given by

$$Q = \begin{bmatrix} p \\ \alpha_v \\ v_i \end{bmatrix}, \Gamma^p = \begin{bmatrix} \alpha_v \frac{\partial \rho_v^\prime}{\partial p} |_{\alpha_v} & \rho_v & 0 \\ \alpha_v \frac{\partial \rho_l^\prime}{\partial p} |_{\alpha_v} & \rho_l & 0 \\ v_i \frac{\partial \rho_v^\prime}{\partial p} |_{\alpha_v} & v_i (\rho_v - \rho_l) & \rho \delta_{ij} \end{bmatrix}.$$ \hspace{1cm} (2.8)

The primed terms can be defined in a number of different ways to render the eigenvalues well-conditioned (Venkateswaran et al., 2002). In these computations the formulation of Venkateswaran and Merkle (1999) is employed.

### 2.2.1 Computational Approach to Multi-stage Hydroturbines

Multi-stage turbomachinery is characterized by having both rotating and stationary stages, and this is no different in hydroturbines. Each stage may consist of a number of blades periodically distributed in a circumferential manner about the axis of rotation of the turbine. Computing the full annulus of each stage, and thus every blade in each stage, can be quite computationally expensive, although it is necessary for a complete unsteady simulation of the machine. However, the distribution of the blades in each stage is usually rotationally symmetric, and to accurately compute steady performance results, only the flow within a single-blade passage of each stage can be simulated while assuming the same flow will occur in all other periodic passages of that stage. These steady-periodic computations...
greatly reduce the computational cost, and it is important to understand how these simulations are conducted.

To accomplish a steady-periodic multiphase simulation it is important to note that all of the computational domains, even the single-passage of the runner blade, which physically rotates in reality, are stationary relative to an inertial (absolute) reference frame. Thus, the rotational effects on the flow are represented by the centrifugal and Coriolis accelerations in the momentum equation component of the source term. Therefore, the absolute source vector $H$ must be replaced with the relative source vector $H_{rel}$, defined as,

$$H_{rel} = \begin{bmatrix} - (\dot{m}^+ + \dot{m}^-) \\ (\dot{m}^+ + \dot{m}^-) \\ \rho \left( g_i - \epsilon_{ijk} \Omega_j \epsilon_{klm} \Omega_l x_m + 2 \epsilon_{ijk} \Omega_j v_k \right) \end{bmatrix}. \quad (2.9)$$

This approach is called a rotating reference frame. The reference frame rotates at rate $\Omega$ about the axis of rotation, where $\epsilon_{ijk}$ is the permutation symbol, and $x_m$ is a component of the position vector. In the hydroturbine case, all stages are stationary except for the runner (blade, hub, and shroud), and thus $\Omega = 0$ for all stages except for the runner. Each stage is solved in its corresponding relative reference frame, hence, the velocities in each stage, are the relative velocities and not the absolute velocities. However, although not shown here, it is simple to transform the runner domain’s velocity field into the absolute velocity field. For the physically stationary stages, where $\Omega = 0$, the relative velocity field is equivalent to the absolute velocity field. Furthermore, the steady-periodic calculations conducted in this dissertation assume incompressible flow. For the mixture approach, this implies constant phasic density (but not necessarily a divergence-free velocity field as mixture density is not constant). This is a valid assumption for steady cavitating flow because incompressible cavitating flow results have been shown to not vary from isothermal compressible cavitating flow results for steady computations (Venkateswaran et al., 2002).

The steady-periodic incompressible multi-phase system of equations, with mass transfer, is defined as

$$\Gamma_{rel}^p \frac{\partial Q_{rel}}{\partial \tau} + \nabla \cdot E_{rel} = H_{rel}. \quad (2.10)$$

Recall that, because steady flow is being considered, the physical time derivative
\( \frac{\partial Q}{\partial t} \) does not appear in these equations. The pseudo-time derivative is still required for convergence purposes, but approaches zero as the solution converges. The variables in the relative pseudo-time vector \((p, \alpha_v, v_i)\) are now the primitive variables which are solved for (where \(v_i\) is now the relative velocity vector. \(Q_{rel}, \mathbf{\Gamma}^p_{rel}, \) and \(E_{rel}\), are unchanged from \(Q, \mathbf{\Gamma}^p, \) and \(E, \) other than the fact that the velocities are now relative velocities, rather than absolute. The mixture density can now be determined from (2.7), as the phasic densities are constant in the incompressible flow, and the phasic equations of state are no longer necessary. Equation (2.10) represents the system of equations solved for the steady-periodic results presented in Chapter 3.

**Periodic Domains**

For steady-periodic computations it is necessary to create a periodic computational domain with periodicity of \(2\pi/n\), where \(n\) is the number of blades in that stage. Thus, to enforce periodicity using a single-blade passage, it is necessary to circumferentially bound the domain with a periodic boundary condition. The periodic boundary condition essentially ensures that scalar values (i.e. pressure, volume fraction, etc.) match at the same \((r, z)\) (radial and axial) location on both boundaries, and that vector quantities (i.e. velocity) maintain their magnitude but have their orientation rotated by \(2\pi/n\) at the same \((r, z)\) location on both boundaries.

It is common for turbomachinery to have a different number of blades between stages, and thus, varying periodicity between stages. Moreover, to help eliminate potential resonance effects from rotor-stator interaction, turbomachines are often designed with a non-integer multiple periodicity between stages. This non-integer multiple periodicity between the guide vane and the runner, for the hydroturbine studied in this dissertation, can be seen in Figure 2.1. Here, the periodic boundaries are labeled and the periodicity of each stage is given.

**Mixing Planes**

From Figure 2.1, it can be seen that the non-multiple periodicity between the vane and runner of the single-blade passage can cause issues at the interface between the stages. It results in an unavoidable, discontinuous solution domain for periodic passages. Furthermore, if a steady solution is calculated with rotating reference
frames, then allowing features such as guide vane wakes to pass through the interface, will result in unphysical solutions. This is because, in reality, the runner will rotate past the wake at certain locations and the wake will not always be in the same location relative to the runner at all times. Consequently, some form of circumferential averaging is required for a coupled, steady solution of multiple stages.

One of the standard procedures for implementing circumferential averaging to compute single-blade passages of multi-stage turbomachines, is to employ mixing planes (Denton and Dawes, 1998; Denton, 1992, 2010). This is the method which is used in this dissertation to accomplish this goal. A mixing plane is a computational interface between the stages which circumferentially averages the flow variables. The mixing planes used in this dissertation are designed to circumferentially average pressure, velocity, and volume fraction. They were built to conserve mass, both radial and angular momentum, and surface averaged pressure across the stage interfaces. The flow-field is then computed in each stage’s domain separately, where the mixing planes act similar to inlets and outlets of the domain, which are updated as the solution converges to steady-state. The flux of mass $\dot{M}$, flux of angular momentum $\dot{P}_\theta$, flux of radial momentum $\dot{P}_r$, and the surface averaged pressure $p_s$.
are written as

\begin{align}
\dot{M} &= \int_S \rho \mathbf{v} \cdot \mathbf{n} dS \\
\dot{P}_\theta &= \int_S r v_\theta \rho \mathbf{v} \cdot \mathbf{n} dS \\
\dot{P}_r &= \int_S v_r \rho \mathbf{v} \cdot \mathbf{n} dS \\
p_s &= \frac{\int_S p dS}{\int_S dS}
\end{align}

(2.11)

where \( S \) is a local surface area (of the mixing plane in this case), \( \mathbf{n} \) is the unit normal vector to \( S \), \( r \) is the radial location, \( v_\theta \) is the fluid angular velocity, and \( v_r \) is the fluid radial velocity. The quantities in (2.11) are equivalent on each side of a mixing plane.

2.2.2 Mass Transfer Modeling

Mass transfer models govern the rate at which fluid changes phases from liquid to vapor and vice versa. In other words, the mass transfer model defines the source terms \( \dot{m}^+ \) and \( \dot{m}^- \) of \( H_{rel} \) in (2.9). There are many different forms of mass transfer models for cavitation and they all have some physical or phenomenological grounding. However, in practice, it has been shown (Morgut and Nobile, 2011; Morgut et al., 2011; Park et al., 2009) that the mass transfer model employed, when optimized for that particular problem with ‘empirical’ coefficients, has little effect on the results. This was further shown to be true for the performance results of cavitating hydroturbines presented by Panov et al. (2012), where 4 different mass transfer models were shown to not have much influence on the performance breakdown of a Francis turbine. In this research, the Kunz model (Kunz et al., 2000) is used, as it is well-proven for simulating cavitating flows, and because any of the other proven models are not expected to influence the results a great deal beyond what the Kunz model predicts. The Kunz mass transfer model is defined as

\begin{align}
\dot{m}^- &= \frac{C_{dest} \rho_v \alpha_l \min [0, p - p_v]}{\frac{1}{2} \rho_\infty v_\infty^2 t_\infty} \\
\dot{m}^+ &= \frac{C_{prod} \rho_v \alpha_l^2 (1 - \alpha_l)}{t_\infty}
\end{align}

(2.12) (2.13)

The destruction of liquid into vapor \( \dot{m}^- \) is similar to the mass transfer terms used in the Merkle model (Merkle et al., 1998) for both destruction and production of liquid. This models the mass transfer rates as proportional to the local liquid
volume fraction and the amount by which the local pressure is below the vapor pressure \( p_v \). For the production of liquid from vapor \( \dot{m}^+ \), the Kunz model takes a different approach by using a simplified version of the Ginzburg-Landau potential (Kunz et al., 1999). The empirical coefficients \( C_{\text{dest}} \) and \( C_{\text{prod}} \) are typically used as modeling parameters for a particular problem, and \( t_\infty \) is a mean flow time scale used to non-dimensionalize the mass transfer rates.

### 2.2.3 Turbulence Modeling

The Boussinesq approximation, which represents the Reynolds stress tensor as a mean viscous stress tensor, is used to allow for an eddy viscosity model. The eddy viscosity model treats turbulence as an increase to the viscosity of the fluid, and the eddy viscosity \( \mu_T \) dominates the molecular viscosity \( \mu \) in highly turbulent regions. The two-equation \( q-\omega_t \) Coakley turbulence model (Marvin and Coakley, 1989) is employed to determine \( \mu_T \) from

\[
\mu_T = \frac{\rho C_\mu q^3}{\omega_t} \tag{2.14}
\]

where \( C_\mu = 0.09 \). \( q \) is the square root of the turbulent kinetic energy and \( \omega_t \) is the specific rate of turbulent dissipation (i.e. the turbulent dissipation over the turbulent kinetic energy). The steady-periodic turbulence model equations that determine \( q \) and \( \omega_t \) at each pseudo-time step are

\[
\frac{\partial Q_T}{\partial \tau} + \nabla \cdot E_{\text{rel}}^T = PROD - DEST \tag{2.15}
\]

\[
Q_T = \begin{bmatrix} \rho q \\ \rho \omega_t \end{bmatrix}
\]

\[
E_{\text{rel}}^T = \begin{bmatrix} \rho q v_j - \left( \mu + \frac{\mu_T}{2} \right) \frac{\partial q}{\partial x_j} \\ \rho \omega_t v_j - \left( \mu + \frac{\mu_T}{2} \right) \frac{\partial \omega_t}{\partial x_j} \end{bmatrix} \tag{2.16}
\]

\[
PROD - DEST = \begin{bmatrix} h_1 \rho \omega_t q \\ h_2 \rho \omega_t^2 \end{bmatrix}
\]

Equation (2.15) is solved in a segregated fashion from the momentum and continuity equations. This two-equation model is quite similar to all of the other two-equation models, such as the standard \( k-\varepsilon \) model. Convection of \( q \) and \( \omega_t \) are represented by
\[ \nabla \cdot \mathbf{E}_{rel}^T \]. The pseudo-time derivative does not require a preconditioning matrix. The term \( PROD - DEST \) is the difference between the production and the destruction of turbulence. This includes a turbulence production term \( P \), which is present in the non-dimensional source function \( h_1 \), and is equivalent to the contraction of the Reynolds stress tensor and the velocity gradient. The non-dimensional source functions \( h_1 \) and \( h_2 \) and their coefficient values are readily available (Marvin and Coakley, 1989).

### 2.2.4 Interface/Shock-Capturing

A multiphase flow can consist of interfaces between the phases. In cavitating flow simulations, variables such as the mixture density can have sharp gradients across these interfaces. The computational resources are usually not available to calculate these interfacial features in fine detail (i.e. with surface-tracking/level-set methods) for industrial flows. Here, the interface is captured with the same methods of shock-capturing that were originally developed for compressible flow simulations. The shock-capturing approach of the interface is implicit in the locally homogenous mixture equations, but the discretization of the flux terms require special attention. Higher resolution upwind schemes, along with flux limiters, which limit the order of accuracy to a 1\textsuperscript{st}-order upwind scheme across interfaces/shocks, but maintain a 3\textsuperscript{rd}-order upwind biased MUSCL scheme in smooth regions of flow, are essential to the shock capturing method. This spatial discretization is discussed in more detail in Section 2.2.5. Shock-capturing maintains an interface without additional calculations, but, the level of resolution of the interface is dependent upon the mesh spacing, and it may be necessary to have a large mesh if a fine level of interfacial detail is required. It should be noted that this approach is not a full Volume of Fluid (VOF) approach, but only pseudo-VOF, as the VOF method requires additional steps (Kinzel, 2008).

### 2.2.5 Spatial Discretization

The choice of spatial discretization is critical to obtaining a stable and accurate numerical solution to partial differential equations. In particular, for fluid mechanics, the discretization of the flux derivative term plays a prominent role in the stability, accuracy, and rate of convergence of the numerical algorithm. Many
different schemes have been developed to discretize the flux term, however, the most commonly employed schemes for CFD can be described by two classes. These classes are upwind differencing (UD) and central differencing (CD).

**Upwind and Central Differencing**

In one-dimension, 1\textsuperscript{st}-order UD discretizes the flux term using a two-point computational stencil, including the point \( i \) and a point ‘upwind’ of \( i \). If information in the problem is propagating from left to right then the upwind point is \( i - 1 \), and in the opposite case the upwind point is \( i + 1 \). 1\textsuperscript{st}-order UD is a Total Variation Diminishing (TVD) scheme, in that it does not produce spurious spatial oscillations of the flow variables (Swanson and Turkel, 1992), even at discontinuities such as shocks or abrupt spatial phase changes (which occur in cavitating flow). However, it can severely dissipate the solution. UD is unconditionally stable if the direction of information propagation is known. If the differencing is done with the wrong direction chosen as upwind, then the scheme is unstable.

A 2\textsuperscript{nd}-order CD scheme in one-dimension employs a two-point computational stencil, using the two points on either side of the point where the derivative is being calculated \((i)\). In this case, that would be \( i - 1 \) and \( i + 1 \). Being 2\textsuperscript{nd}-order, CD is by definition, more formally accurate than 1\textsuperscript{st}-order UD. Nevertheless, CD is not TVD, and spurious oscillations of flow variables, which are physically inaccurate and unstable, can occur. This is especially true in regions with discontinuities. However, CD provides more accurate solutions than UD in smooth regions because it does not dissipate the solution nearly as much as UD.

Both UD and CD have their benefits and disadvantages for particular flow problems. In this dissertation, to obtain a cavitating flow field in a hydroturbine, it is beneficial to employ 1\textsuperscript{st}-order UD in locations where sharp discontinuities are present \((i.e.\) across phase interfaces), but maintain the accuracy of higher-order schemes in regions where the flow variables are smooth \((i.e.\) the majority of cells in the computational domain). To do so, ‘higher resolution’ schemes are used to produce high-order UD which tends to 1\textsuperscript{st}-order UD with the use of a flux-limiter \((a\ variable\ which\ limits\ the\ solution\ gradient\ near\ discontinuities)\).

In using basic UD (and its higher resolution counterparts), it is vital for the scheme to be adaptive and determine the direction of information propagation at any point in the computational domain, to ensure the discretization is upwind.
The Method Of Characteristics (MOC) suggests that each particular eigenvalue component of the system should be differenced according to the sign of the eigenvalue. In the Euler and Navier-Stokes equations, the MOC shows that information can propagate in multiple directions, concurrently, arising through the appearance of both positive and negative eigenvalues (in a simple one-dimensional problem). Thus, to ensure stability, it is necessary for the scheme to allow for UD in multiple directions, simultaneously. To accomplish this, the flux must be split into two parts, corresponding to both possible directions of information propagation.

**Flux Splitting**

As previously stated, the purpose of flux splitting is to allow for UD in a system where characteristics propagate in both directions. Here, only one-dimensional examples are considered. There are two common procedures for splitting the flux and they are flux-vector splitting and flux-difference splitting. Flux-vector splitting is not general, in that it is inconsistent when the flux vector is non-homogeneous. A non-homogeneous flux vector implies that $\mathbf{E} \neq \mathbf{A}\mathbf{Q}$, where $\mathbf{E}$ is the flux vector, $\mathbf{A}$ is the flux Jacobian matrix defined by $\mathbf{A} = \frac{\partial \mathbf{E}}{\partial \mathbf{Q}}$, and $\mathbf{Q}$ is the vector of dependent variables. In fluid problems, non-homogeneous flux vectors can arise. Flux-difference splitting allows for both homogeneous and non-homogeneous flux vectors.

In the same manner that 1st-order UD of a scalar quantity can be interpreted as 2nd-order CD with an additional scalar artificial dissipation term, 1st order UD of the flux vector can be interpreted as 2nd-order CD of the flux vector with an additional ‘matrix dissipation’ term (Merkle, 2009). Flux-difference splitting accomplishes this by first discretizing the flux derivative in a general manner such that

$$\frac{\partial \mathbf{E}}{\partial x} = \frac{\tilde{\mathbf{E}}_{i+1/2} - \tilde{\mathbf{E}}_{i-1/2}}{\Delta x}. \quad (2.17)$$

Here, $\tilde{\mathbf{E}}$ represents the ‘numerical’ flux which is evaluated at a cell face location of cell $i$. The order of differentiation is dependent upon how $\tilde{\mathbf{E}}$ is defined. For a choice of numerical fluxes for 1st-order UD, the following can be recovered (and similarly an expression for $\tilde{\mathbf{E}}_{i-1/2}$).

$$\tilde{\mathbf{E}}_{i+1/2} = \frac{1}{2} (\mathbf{E}_i + \mathbf{E}_{i+1}) - \frac{1}{2} \left( \delta \tilde{\mathbf{E}}_{i+1/2}^+ - \delta \tilde{\mathbf{E}}_{i+1/2}^- \right)$$

$$= \frac{1}{2} (\mathbf{E}_i + \mathbf{E}_{i+1}) - \frac{1}{2} |\mathbf{A}_{i+1/2}| \delta \mathbf{Q}_{i+1/2} \quad (2.18)$$
The first term will provide 2nd-order CD when used in (2.17). The second term is made up of the numerical flux vectors, split by their characteristic directions of propagation (+ and − for right-running flux and left-running flux, respectively). The variable $\delta$ in front of a cell face vector represents the difference between the adjacent cell center values of that vector, for example $\delta \tilde{E}_{i+1/2} = \tilde{E}_{i+1} - \tilde{E}_{i}$. As shown, this equation can be reformulated in terms of the numerical flux Jacobian positive-definite matrix dissipation $|A_{i+1/2}| = A_{i+1/2}^+ - A_{i+1/2}^-$. In this approach it is clear that a 1st-order UD scheme is equivalent to a 2nd-order CD scheme with additional matrix dissipation.

The order of the upwind scheme can then be controlled by generalizing the dissipation term to

$$
\tilde{E}_{i+1/2} = \frac{1}{2} (E_i + E_{i+1}) - \tilde{D}_{i+1/2} \\
\tilde{D}_{i+1/2} = k_1 \left( \delta \tilde{E}_{i+1/2}^+ - \delta \tilde{E}_{i+1/2}^- \right) - k_2 \left( \delta \tilde{E}_{i-1/2}^- - \delta \tilde{E}_{i+3/2}^- \right)
$$

(2.19)

This dissipation term allows for the control of the order of accuracy of the UD, while ensuring UD through flux splitting. For 2nd-order CD $k_1 = 0$ and $k_2 = 0$, while if 1st-order UD is desired, then $k_1 = \frac{1}{2}$ and $k_2 = 0$. Third-order upwind biased differencing can be recovered when $k_1 = k_2 = \frac{1}{6}$.

**Higher Resolution Schemes for Upwind Differencing**

To develop and solve a discretized Riemann problem (essentially a hyperbolic equation along with piecewise constant data and a jump discontinuity (LeVeque, 2002)), Godunov (1959) used a series of cells with piecewise constant data across each cell. Thus, at each cell face a discontinuity could occur and Godunov’s approach could be considered to be representative of a series of Riemann problems. The UD schemes which arose from this fully non-linear solution were only first order, because of the piecewise constant data across any cell. To develop high resolution schemes for UD, approximate (i.e. linear) Riemann solvers were developed by treating the dependent variable vector $Q$ across the cell as linear and quadratic functions, allowing for the recovery of higher-order UD schemes. To accomplish this, cell face values such as $Q_{i+1/2}^L$ and $Q_{i+1/2}^R$ are determined from a reconstructed continuous function $Q(x)$. The superscript of $L$ and $R$ imply that $Q_{i+1/2}$ is evaluated on either side of the cell face between $i$ and $i + 1$, and because this is a Riemann...
problem, these values may be different. \( Q(x) \) can be chosen to be a constant, linear, or quadratic function depending on the order of accuracy needed. \( Q(x) \) is found by using a series of adjacent cell center average values \( (\bar{Q}_{i-1}, \bar{Q}_i, \text{ and } \bar{Q}_{i+1}) \) to construct a continuous function based on the slopes and curvatures of the averaged values (Merkle, 2009).

The result from the reconstruction problem across a cell interface can be described with the following equations for the primitive flow vector on either side of the interface:

\[
\begin{align*}
Q^L_{i+1/2} &= \bar{Q}_i + \frac{\kappa_1}{4} (1 + \kappa_2) (\bar{Q}_{i+1} - \bar{Q}_i) + \frac{\kappa_1}{4} (1 - \kappa_2) (\bar{Q}_i - \bar{Q}_{i-1}) \\
Q^R_{i+1/2} &= \bar{Q}_{i+1} - \frac{\kappa_1}{4} (1 - \kappa_2) (\bar{Q}_{i+2} - \bar{Q}_{i+1}) - \frac{\kappa_1}{4} (1 + \kappa_2) (\bar{Q}_{i+1} - \bar{Q}_i)
\end{align*}
\] (2.20)

This higher resolution scheme can be different orders of accuracy depending on the choices of variables \( \kappa_1 \) and \( \kappa_2 \). For example, \( \kappa_1 = \kappa_2 = 0 \) gives 1\textsuperscript{st}-order UD, \( \kappa_1 = 1 \) and \( \kappa_2 = 0 \) gives third-order upwind biased, while \( \kappa_1 = \kappa_2 = 1 \) gives 2\textsuperscript{nd}-order CD. This approach, where the function \( Q(x) \) is reconstructed in this manner is called MUSCL (Monotonic Upstream-Centered Scheme for Conservation Laws), and was first developed by Van Leer (1979).

The final step is to calculate the numerical fluxes through the cell faces, from the \( Q^L \) and \( Q^R \) values. The jump across the cell face is evaluated by taking the difference of \( Q^R \) and \( Q^L \). The numerical flux can be related to this jump with a conservative method called Roe flux difference splitting (Roe, 1981) which is shown below.

\[
\tilde{E}_{i+1/2} = \frac{1}{2} (E_i + E_{i+1}) - \frac{1}{2} |\tilde{A}_{i+1/2}| (Q^R_{i+1/2} - Q^L_{i+1/2})
\] (2.21)

Roe flux difference splitting obtains the numerical flux at the cell face, in a manner that decomposes the flux into its characteristic directions, while satisfying conservation of the numerical flux. Conservation is ensured by arithmetic averaging when obtaining the ‘magnitude’ of the averaged numerical flux Jacobian matrix at the cell face \( \tilde{A}_{i+1/2} \), which is a function of \( Q^L_{i+1/2} \) and \( Q^R_{i+1/2} \).

### 2.2.6 Steady-Periodic Solution Strategy

A modified version of the UNCLE CFD solver (Taylor et al., 1995), named UNCLE-M, is used to solve the steady-periodic cavitating flow field in the hydroturbine. UNCLE-M extends the original parallel, block-structured, finite-volume solver to
include multiphase and compressible capabilities, as well as mass transfer. The multiphase implementation uses the constant phasic density, locally homogenous mixture approach previously described. The primitive variable vector, at the cell faces, is calculated with a 3rd-order upwind-biased MUSCL scheme. The MUSCL scheme is limited by van-Albada flux-limiters to limit the solution to 1st-order UD across sharp gradients, but maintain the accuracy of 3rd-order upwind biased differencing in regions of spatially smooth flow. Roe’s flux-difference splitting scheme is used to calculate the numerical flux vector at the cell faces from the primitive variable vector at the cell faces. Arithmetic averaging is used to determine the flux-vector Jacobians at the cell face, which is necessary to conserve the numerical fluxes. The flux-Jacobians at the cell-centers are formed with both analytical and numerical methods. The segregated turbulence model equations are solved to 1st-order accuracy. A block-symmetric Gauss-Seidel iteration procedure is employed for the implicit integration of the pseudo-time, to obtain the primitive variables at the next pseudo-time step. This iteration procedure continues until convergence is reached. It is important to note that before this entire solution procedure begins, the body-fitted computational mesh must first be transformed to a Cartesian computational domain.

2.3 Details of Francis Hydroturbine Case

The Francis hydroturbine geometry, which is used to obtain the multiphase CFD results presented in this dissertation, was provided by industry partner Weir American Hydro (WAH). This particular machine is quite old and was a refurbishment project of WAH. The plant is a low-head run-of-river facility. The hydroturbine consists of a penstock, partial spiral casing, stay vane stage, guide vane stage, runner stage, and elbow draft tube diffuser. The refurbishment was to the runner stage, and the geometry studied here was one of the candidate runners to replace the old runner.

2.3.1 Geometry

The hydroturbine consists of 4 inlets into the penstock. 3 of these inlets converge into what is known as a partial spiral casing, because it does not fully enshroud
the stay vanes. Partial spiral casings are quite common for low-head run-of-river turbines, and even though oddly configured, they are designed to provide a similar flow field at all circumferential locations entering the guide vanes. The flow entering through the other inlet is swirled slightly by the penstock geometry, but mostly enters directly into the stay vane stage.

The stay vanes have a periodicity of $2\pi/n$ where $n = 20$. However, although the stay vanes have this periodicity, there are not 20 stay vanes, but only 18. This is because the penstock/partial spiral casing walls extend into the stay vane stage at two locations, and the walls themselves support the structure and direct the flow into the guide vane stage. Thus, two stay vanes are absent, although they maintain the periodicity as if there were 20 vanes.

The guide vane stage (see Figure 2.2) consists of $n = 20$ guide vanes with a periodicity similar to the stay vanes. At many guide vane opening angles, the trailing edge of the vane hangs over the shroud of the runner. This geometrical characteristic is considered in the CFD and is shown to produce cavitation issues at practically all simulated Thoma numbers. In this dissertation, the guide vanes are set to an opening angle of $29.0^\circ$, which is close to the BEP of the turbine.
There are a total of \( n = 11 \) runner blades in this machine, and thus the runner has a different periodicity than the guide vanes. These blades are attached to the hub and shroud, and all three components rotate together at rate \( \Omega \). Following the runner is the elbow draft tube. It is clear from Figure 2.2 that at the exit of the runner, and entrance to the draft tube, there is an abrupt geometric expansion. Following the expansion the flow does not have much of a conical diffusion region, as the elbow begins almost immediately and is a rather sharp redirection of the flow. The draft tube has a circular inlet and an oblong outlet.

Refurbishing this turbine with a new runner provided a difficult challenge to the WAH engineers. As mentioned, due to the high cost of modifying components such as the penstock, spiral casing, and draft tube, modern refurbished runners are designed to be installed without modifying those components. In this case, the WAH engineers remarked that this particular draft tube was ‘the worst the company had ever seen.’ They also indicated that with the modern runner, most of the losses occurred in the draft tube.

### 2.3.2 Computational Mesh

To allow for the assumption of periodicity and to reduce computational cost, the penstock, partial spiral casing, and stay vanes are not included in the steady-periodic calculations. The inlet to the computation is chosen to be a surface of constant radius, just upstream of the periodic, single-passage guide vane. A periodic single-blade passage was meshed for the runner region as well. The full 360° draft tube was meshed and included in the steady-periodic computations even though it is not an axisymmetric component of the machine. Assuming an axisymmetric draft tube in the CFD would not allow for the machine losses to be captured correctly, hence, the full, poor performing draft tube must be included. The outlet to the computation is a plane at the exit of the draft tube.

The solver, UNCLE-M, requires a fully-structured mesh of hexahedra. It places some further constraints on the mesh at the periodic and mixing plane boundaries. The periodic boundaries are required to be point-matched, meaning that this boundary must have exactly the same surface mesh on either side of the single-blade passage. The mixing plane boundary is required to have the same number of cells in the circumferential and blade span-wise directions, on both sides of the
Table 2.1: Number of cells within each periodic stage for the coarse and fine meshes.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Guide Vane</th>
<th>Runner</th>
<th>Draft Tube</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>1.5 × 10^6</td>
<td>2.2 × 10^6</td>
<td>1.4 × 10^6</td>
<td>5.1 × 10^6</td>
</tr>
<tr>
<td>Medium</td>
<td>2.8 × 10^6</td>
<td>3.7 × 10^6</td>
<td>2.0 × 10^6</td>
<td>8.5 × 10^6</td>
</tr>
<tr>
<td>Fine</td>
<td>4.6 × 10^6</td>
<td>6.5 × 10^6</td>
<td>3.1 × 10^6</td>
<td>14.2 × 10^6</td>
</tr>
</tbody>
</table>

interface. The cells do not have to line up circumferentially on both sides of the interface, but the circumferential grid lines are required to match at each span-wise location on both sides of the interface.

Three grids were created, using the grid generation software Pointwise (Pointwise, 2011), to allow for a mesh refinement study. The same boundary layer spacing was used for each level of refinement because all of these meshes were employing wall functions, regardless of the refinement. Further, even for the coarse mesh, the number and distribution of grid points around the profile of each blade was determined by the rules-of-thumb for hydrofoil shapes presented by Chan et al. (2002). Thus, the grid point distribution around the hydrofoil at a given span-wise location was not refined further for the more refined meshes. Therefore, each level of refinement consisted of increasing the number of cells in both the circumferential direction, and the blade span-wise direction. The cell count for each mesh can be seen in Table 2.1.

The mesh was first created for the full-scale prototype simulations. The reduced-scale model can be simulated as well, by simply reducing the dynamic molecular viscosity to match the experimental Reynolds number, and reducing gravity to match the experimental Froude number. Thus, the same geometry, mesh, and flow conditions are used for the model simulations as are used for the prototype, except that viscosity and gravity are modified to reach the model experiment conditions. Since the same mesh is used for both scales, the closer wall spacing in the model allows for better resolution of the boundary layers, however, the boundary layers are still adequately resolved to use wall-functions in the prototype as well. The boundary layer resolution and the scaling will be discussed further in Sections 2.3.3 and 2.4.

The fine guide vane mesh is shown in Figure 2.3. The guide vane has an overhang on its lower end, which extends above the runner as the end wall begins to curve
downwards into the runner shroud. The gap beneath the overhanging guide vane is prone to cavitation and was meshed to capture these effects. Close-up images of the leading-edge shroud side, and the trailing-edge shroud side of the fine runner mesh are displayed in Figure 2.4. The fine draft tube mesh (Figure 2.5) also contains the hub nose cone. The nose cone was included in the draft tube domain to allow the non-axisymmetric effects of the hub vortex rope to be captured in the periodic simulations. Figure 2.5 also shows a slice of the mesh beneath the nose cone.
2.3.3 Boundary Conditions

Prescribing the boundary conditions (BC) is essential to solving any partial differential equation(s) (PDE) or system of PDE. Furthermore, because the solution is critically dependent upon the BC, correctly defining which conditions are used, and the input values, is paramount to the numerical procedure’s convergence and results. Two primary types of BC are employed in CFD, and these are called Dirichlet and Neumann. A Dirichlet BC prescribes a fixed value to a variable, such as $v = \text{constant}$ at an inlet and a no-slip wall condition, or $p = \text{constant}$ for a pressure outlet. A Neumann BC enforces the gradient of a variable, with respect to boundary normal, to be a fixed value, such as $\frac{\partial v}{\partial n} = \text{constant}$ at a pressure outlet, or $\frac{\partial p}{\partial n} = \text{constant}$ for a velocity inlet and a no-slip wall condition.

The Francis turbine is simulated with a fixed-value total pressure inlet, based on the net specific energy of the machine, upstream of the guide vane leading edge. The angle of the flow is also chosen so that it is incident with the leading edge of the guide vane. The velocity magnitude is prescribed a zero-valued Neumann condition, while the vapor volume fraction and turbulence quantities are Dirichlet conditions.

The outlet, at the exit of the draft tube, is given a Dirichlet condition of $p = 0$ at a mean altitude of the exit. It should be noted that the hydrostatic pressure gradient is imposed on both the inlet and the outlet boundaries. Thus, it is necessary to set the pressure value on the boundary at a certain vertical position. A Neumann condition is prescribed for the velocity, and Dirichlet conditions are input for the
turbulence variables and the vapor volume fraction. The outlet is also a hybrid outlet. In cells where there is back-flow into the domain, the pressure becomes Neumann, and the other variables Dirichlet. Using the hybrid outlet allows for improved computational stability.

At the physical walls in the domain, all of the scalar variables are assigned a zero-valued Neumann condition, as there is no flux of these quantities through the walls. The relative velocity is given a zero-valued Dirichlet condition, to impose the no-slip condition. It is important that the relative velocity satisfies this condition because, if the reference frame of the stage is rotating, then it is the relative velocity at the wall which is zero and not the absolute velocity. The turbulent boundary layers are resolved to a level such that employing wall functions is necessary, and thus, the force on a wall (which is used as a boundary condition in the momentum equation) is determined using the logarithmic law of the wall. To use wall functions, the dimensionless distance of every first cell center point from the wall \( y^+ \) must lie approximately between 30 and 300 (where the upper bound is extended as Reynolds number increases). \( y^+ \) is determined from the following

\[
y^+ = \frac{d \rho u_{\tau}}{\mu}
\]

where the friction velocity \( u_{\tau} = \sqrt{\tau_w/\rho} \), the wall shear stress is \( \tau_w = \mu \frac{du}{dy} \), \( d \) is the distance from the wall, \( u \) is the velocity component tangent to the wall located at the cell center, and \( y \) is the direction normal to the wall. The law of the wall for the dimensionless velocity is

\[
u^+ = \frac{1}{\kappa} \log y^+ + B
\]

where \( \kappa \) and \( B \) are known constants. Finally, the dimensional velocity is related to the dimensionless velocity with \( u = u^+ u_{\tau} \). The values of \( y^+ \) for the converged steady simulations of the model and prototype can be seen in Table 2.2. Since the same mesh is used for the model and prototype, and \( y^+ \) is Reynolds number dependent, it is clear why the values of \( y^+ \) are lower for the model. In some cases the values are below 30, however, the blended wall functions of Spalding (Shih et al., 1999) are used to accurately predict \( u \), by changing the form of the equation for \( u^+ \) when \( y^+ \) does not lie in the log layer.
Table 2.2: Average, minimum, and maximum values of $y^+$ of each stage for the model and prototype scales.

<table>
<thead>
<tr>
<th>Scale</th>
<th>Component</th>
<th>$y_{avg}$</th>
<th>$y_{min}$</th>
<th>$y_{max}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
<td>Guide Vane</td>
<td>7.30</td>
<td>0.36</td>
<td>42.6</td>
</tr>
<tr>
<td>Model</td>
<td>Runner Blade</td>
<td>3.18</td>
<td>0.19</td>
<td>10.9</td>
</tr>
<tr>
<td>Model</td>
<td>Draft Tube</td>
<td>3.47</td>
<td>0.38</td>
<td>7.64</td>
</tr>
<tr>
<td>Prototype</td>
<td>Guide Vane</td>
<td>94.3</td>
<td>7.16</td>
<td>458</td>
</tr>
<tr>
<td>Prototype</td>
<td>Runner Blade</td>
<td>39.3</td>
<td>1.91</td>
<td>126</td>
</tr>
<tr>
<td>Prototype</td>
<td>Draft Tube</td>
<td>44.4</td>
<td>1.97</td>
<td>95.2</td>
</tr>
</tbody>
</table>

Figure 2.6: Periodic boundaries bound the mesh circumferentially while mixing planes define the interface between each stage.

Periodic boundary conditions are used for the single-blade passages. They were previously discussed in Section 2.2.1. Also, mixing planes are employed between all stages of the hydroturbine. These were discussed in Section 2.2.1. Both the periodic boundaries and the mixing planes can be seen in Figure 2.6. All of the boundary conditions are implemented into the finite-volume formulation through the ghost cell approach (see LeVeque (2002) for discussion on ghost cells).

2.3.4 Initial Conditions and Convergence

The single-phase steady-periodic computations are initialized by prescribing the same velocity field, and outlet pressure conditions, across the entire domain of internal cells. To start, a large artificial compressibility $\beta$ and first-order upwind schemes are used. $\beta$, which is related to $\frac{\partial \rho'}{\partial p'}$ in the preconditioning matrix $\Gamma^p$, acts to decouple the pressure and velocity fields allowing for more stability in the
initialization stages. After initialization, the numerical fluxes are modified to be calculated with 3rd-order flux-limited MUSCL schemes. As the solution becomes more stable with successive iterations, the artificial compressibility is gradually lowered until $\beta$ is roughly $(10v_\infty)^2$ (Kinzel, 2008), where $v_\infty$ is the magnitude of the velocity of the runner at discharge radius $R$. At this point, the preconditioning terms are considered to have a negligible effect on the solution. Next, the cavitation modeling is switched on for the multi-phase calculation, and this simulation is run until convergence.

The residuals of the pressure, velocity components, turbulence variables, and volume fraction, are all converged more than 3 orders of magnitude for each case. Furthermore, the simulations are run until the values of torque and flow rate hardly change over a few thousand iterations. At this point the cases are considered to be converged and ready for post-processing.

2.3.5 Computational Resources

Each steady-periodic cavitating CFD solution is computed on the Department of Defense High Performance Modernization Program (DoD HPCMP) SGI Ice X (1.5 PFLOPS) named Spirit, belonging to the U.S. Air Force Research Laboratory (AFRL). Spirit contains 4590 compute nodes with 32 GB of memory each. Each node has 16 Intel E5 Sandy Bridge cores, each with a speed of 2.6 GHz. Each case is computed on 160 cores (10 nodes), and takes approximately 60 hours (2.5 days) to ensure convergence with the finest mesh, after the cavitation modeling is switched on. It should be noted that the UNCLE-M solver is relatively old and much slower when compared with more modern CFD solvers (such as the commercial solver STAR-CCM+ which is utilized to compute the unsteady results; see Chapters 4 and 5). However, at this time, STAR-CCM+ does not have the ability to use mixing planes with multiphase flow.

2.4 Model Experiments

The testing of reduced-scale models of hydraulic turbines is an established process regulated by an international standard (Dörrler et al., 2013; IEC, 1999). The model-scale experiments for the Francis turbine simulated in this dissertation were
conducted at the Turboinstitut in Slovenia, in a closed test loop. The prototype-to-model geometrical scaling ratio was 14.5:1. The model tested, was geometrically similar to the prototype from the penstock inlet to the draft tube outlet (the entire flow path through the machine). However, the fluid dynamic similarity between the model and prototype is never exact. The Reynolds number

\[
Re = \frac{\rho_l \Omega \cdot RD}{\mu}
\]

is usually much smaller in the model. Note that \( D \) and \( R \) are the diameter and radius of the runner exit, and \( \Omega \) is the rotation speed of the runner. The smaller \( Re \) results in thicker boundary layers, and thus a lower power and efficiency in the model than would be expected in the higher \( Re \) prototype. Experience has provided standard empirical relations (IEC, 1999) which are used to predict the increase in the prototype power and efficiency from the model scale results. The standard is to apply the empirical relations to the BEP model results. The boosts to efficiency and power at BEP are then assumed to be the same across all design points, and even for the cavitation results. The dimensionless power coefficient \( P^* \) and efficiency \( \eta \) are given by

\[
P^* = \frac{2P}{\pi \rho_l \Omega^3 R^5}
\]

\[
\eta = \frac{P}{\rho_l Q \cdot E}
\]

Here, \( P \) is the power, defined as the product of the torque on the shaft \( T \) and the rotation speed of the runner \( (P = T \cdot \Omega) \). \( Q \) is the volumetric flow rate and \( E \) is the net specific energy across the machine, which is defined as

\[
E = \left[ \frac{p_{abs,1}}{\rho_l} + gZ_1 + \frac{v_1^2}{2} \right] - \left[ \frac{p_{abs,2}}{\rho_l} + gZ_2 + \frac{v_2^2}{2} \right]
\]

where \( p_{abs}, Z, \) and \( v \) are the absolute pressure, elevation, and velocity, defined at the machine inlet and outlet locations denoted by the subscripts 1 and 2, respectively, and depicted in Figure 2.7.

Unlike the Reynolds number, the dimensionless flow coefficient \( \phi \) and specific energy coefficient \( \psi \)

\[
\phi = \frac{Q}{\pi \Omega R^5}
\]

\[
\psi = \frac{E}{Q \cdot \pi \Omega R^5}
\]
Figure 2.7: Generic run-of-river plant with upstream location 1 and downstream location 2 at the exit of the draft tube (Avellan, 2004).

\[ \psi = \frac{2E}{\Omega^2 R^2} \]  

(2.29)

are considered similar for both scales (note that (2.26) can also be written as \( \eta = \frac{P^*}{\phi \psi} \)). Moreover, for cavitation tests, the Thoma number

\[ \sigma = \frac{NPSE}{E} \]  

(2.30)

is considered to be similar at both scales. \( NPSE \) is the Net Positive specific Suction Energy given by

\[ NPSE = \left[ \frac{p_{abs,2}}{\rho_l} + gZ_2 + \frac{v_2^2}{2} \right] - \frac{p_v}{\rho_l} - gZ_r \]  

(2.31)

where \( p_v \) is the vapor pressure and \( Z_r \) is the elevation of a chosen reference location. It is strongly recommended (Avellan, 2004) to define the elevation of the lowest pressure to be \( Z_r \), in a best attempt to overcome gravitational scaling effects. Cavitation tests are conducted by maintaining a constant \( \psi \), and guide vane opening angle, while gradually lowering \( \sigma \). This is done by maintaining constant \( E \) but modifying the test loop pressure with a vacuum pump, and thus lowering \( NPSE \). As \( \sigma \) decreases and cavitation occurs to a greater extent, \( \phi \), \( P^* \), and \( \eta \) reach a critical point where they begin to decrease. When the efficiency has dropped by greater than 0.5%, the test is usually concluded, with the main result being the critical Thoma number for which cavitation begins performance breakdown of the turbine. Further lowering of \( \sigma \) is not expected to provide results which are scalable.
to the prototype. The reason is that there is a lack of Froude similarity between the two scales. Lack of Froude similitude, where Froude number $Fr$ is defined by

$$Fr = \sqrt{\frac{E}{gD}},$$  \hspace{1cm} (2.32)

is caused by the inability to scale the hydrostatic pressure field, caused by the gravitational field, across the machine. Once cavitation begins to have a large effect on the performance, the model results are not reliable for predicting the prototype performance behavior by using the standard efficiency increase estimated at BEP in non-cavitation tests. This aspect of cavitation model testing is a great hindrance to understanding the cavitation characteristics of a full-scale machine. CFD can easily compute both scales at lower-than-critical Thoma numbers, and provide an understanding of the differences between the scales caused by lack of Froude similitude.

The computational study presented in this dissertation is compared to 4 cases of the experimental model cavitation tests. These cases are at a constant guide vane opening angle close to BEP. The variable $\sigma$ is lowered in each case as the turbine transitions to performance breakdown (Figure 2.8). From the model experiments, using the standard, an approximately 4.4\% increase in efficiency is predicted for the prototype, and this is shown in Figure 2.8. This is the predicted increase in the efficiency due to lack of $Re$ similitude. The model tests for this particular turbine are at $Re \approx 7 \times 10^6$ while the prototype is at $Re \approx 1 \times 10^7$.

The input values of the simulations are obtained from the experimental values of $\psi$ and $\sigma$, for both scales. Since the full-scale mesh geometry is used to calculate the prototype and the model CFD solutions, the viscosity $\mu$ and gravity $g$ are altered for the model-scale simulation to match the $Re$ and the $Fr$ number of the model tests. This results in a model-simulation viscosity which is 2 orders of magnitude larger than the viscosity of water for the prototype. Furthermore, the gravity in the model-simulation is 1 order of magnitude smaller than the gravity of the prototype. The ratios of model to prototype $Re$ and $Fr$ are roughly 0.07 and 3.6, respectively.

A constant zero-pressure outlet is maintained at the exit of the draft tube for all computational cases. Therefore, to modify $\sigma$, the $NPSE$ is changed by altering the $p_v$ accordingly. In reality, $p_v$ is a material constant at a particular temperature and $\sigma$ is not varied in this way. However, in the computations it is much simpler to
Figure 2.8: Efficiency $\eta$ versus Thoma number $\sigma$ for all four cases of the model experiment, along with the industry standard scaled efficiency estimate of the prototype for each $\sigma$. Images of hub vortex cavitation from the model tests are shown at their corresponding $\sigma$.

keep the outlet pressure at zero, and simply raise $p_v$ in the correct manner, to lower $\sigma$. Again, the physical model experiments require a vacuum pump to lower $\sigma$. 
3 Steady-Periodic Results and Discussion

3.1 Introduction

Steady-periodic multiphase Computational Fluid Dynamics (CFD) simulations are conducted to capture cavitation breakdown in a Francis hydroturbine due to large-scale vaporous structures. A reduced-scale model and a full-scale prototype are investigated to display differences in vapor content and machine performance associated with lack of Reynolds ($Re$) and Froude ($Fr$) similarity. Furthermore, the performance of each machine stage, at both scales, is quantitatively analyzed at different Thoma numbers ($\sigma$) along the cavitation breakdown curve. These two significant and novel contributions help extend the knowledge of cavitating flow performance in the hydropower community. In addition, a formal mesh refinement study is conducted on machine efficiency ($\eta$) and volume of vapor, with 3 mesh levels and Richardson extrapolation, to ensure convergence.

Reduced-scale model tests present a unique challenge to cavitation studies in hydroturbines. Since model tests are much smaller than the full-scale prototype, there is no $Fr$ similarity between the scales (Avellan, 2004). For single phase flow this is not an issue, but this lack of similitude can cause dissimilar vapor structures and performance effects between scales. For this reason, and in the absence of scaling laws for these effects, model based guarantees of performance are restricted to $\sigma$ values greater than those corresponding to a 0.5% drop in $\eta$ (IEC, 1999). Thus, only small cavitation structures are considered useful in model testing, limiting the effectiveness of model tests.

Although there is a great deal of CFD research focused on hydroturbines, and a wealth of CFD analyses of cavitating flow in turbomachinery, investigations of hydroturbines utilizing multiphase CFD are rather scarce. Published cavitating
hydroturbine CFD studies have focused either on steady-state performance breakdown across a range of $\sigma$ values (Lipej et al., 2002; Liu et al., 2009, 2005; Panov et al., 2012), or cavitation effects on unsteadiness (Guo et al., 2007; Jošt and Lipej, 2009; Wu et al., 2011). Results of these investigations compared well with physical tests. However, all were either at model or prototype scale. Thus, differences between model and prototype cavitation breakdown were not directly compared. Recently, Panov et al. (2014) presented a study of unsteady cavitating model and prototype machines. Nevertheless, steady-state performance was not investigated, and both scales had different runner geometries; consequently their results could not be directly compared across scales in an adequate manner. Stage-by-stage performance decompositions have been published with single-phase CFD results for the entire operating range of a Francis turbine (Vu and Retieb, 2002), and for various flow rates of a Francis turbine (Jošt and Lipej, 2009). However, this dissertation marks the first time this type of analysis is presented for cavitating flow in a hydroturbine. Furthermore, while some of the cavitating CFD hydroturbine studies were supported with mesh convergence, most were only completed with 2 levels of refinement and using torque as the convergence parameter rather than a more comprehensive quantity, such as $\eta$. In addition, another performance parameter should be monitored, corresponding to convergence of multiphase flow.

In this chapter, mixing planes are used to couple rotating and stationary stages of the turbine yielding a coupled solution for the guide vanes, runner blades, and draft tube. Initially, single-phase results are presented along with a mesh convergence study. Next, the cavitating machine performance results are shown alongside a multiphase mesh convergence study. The multiphase mixture approach is used to simulate the fluid dynamics, and the Kunz cavitation model is employed to govern the mass transfer between liquid and gas phases (Kunz et al., 2000). The solutions are compared with experimental results across a range of $\sigma$ values, displaying the major cavitation features in the machine. Furthermore, $\sigma$ is reduced below the experimental values to clearly, computationally distinguish model and prototype behavior. Excellent mesh convergence is obtained, using machine efficiency and volume of vapor as variables of interest. Finally, a stage-by-stage performance decomposition is presented for all Thoma number conditions, and for both scales, providing insight on the specific locations of losses in a hydroturbine during cavitation breakdown.
In addition to the quantitative results, this chapter displays the qualitative features of hydroturbine single-phase flow and cavitation. Pressure/velocity contours and streamlines are given for the single-phase results. Isosurfaces of vapor volume fraction (representing the cavitation structures) are displayed for the multiphase results and are shown alongside drawings and photos from the experiments. The post-processing software that produced these flow visualization images was Fieldview (Intelligent-Light, 2013).

### 3.2 Single-Phase Flow

Initially, the model and prototype are converged for single-phase flow. Most of the discussion in this chapter will focus on the cavitation simulations. However, it is worthwhile to show some results of the single-phase pressure and velocity fields to get a sense for the flow through the turbine before vapor begins to form. First, the results of the mesh convergence study for the single-phase cases are given. Next, the model-scale single-phase images are shown and discussed.

#### 3.2.1 Mesh Convergence Study

To ensure convergence of a variable of interest, it is necessary to conduct a formal mesh convergence study for that variable. In this chapter, the steady-state performance of the hydroturbine is computed and analyzed. Thus, for single-phase flow, the primary quantity of interest is the machine efficiency $\eta$. If $\eta$ is shown to converge, then the other global/integrated quantities which make up $\eta$, such as flow coefficient and power coefficient (runner torque), are implicitly shown to converge as well. The specific energy coefficient is set to be equivalent between the model and prototype, as is common practice. Here, the single-phase results are computed at the same guide vane opening angle as the cavitating solutions are computed at. Consequently, these results can be viewed as operating at a high enough $\sigma$ that no cavitation forms in the machine.

The standard method to quantify mesh convergence is with a procedure known as Richardson extrapolation. This procedure is outlined in detail by Roache (1994, 1998). This technique requires that the same numerical method be used for each level of mesh refinement, and that the variable of interest is within the
asymptotic range of convergence (Roache, 1998). First, a representative cell size $h_{cell}$ is determined for each mesh, based on the volume of the $i^{th}$ cell $\Delta V_i$ and the total number of cells in that mesh $N$ (3.1). Now, let $h_{cell_1} < h_{cell_2} < h_{cell_3}$, such that 1 represents the fine mesh, 2 the medium, and 3 the coarse mesh. Grid refinement ratios $r_{21}$ and $r_{32}$ are defined as in (3.2). If $r_{21} \neq r_{32}$ then an iterative scheme is necessary to complete the extrapolation (Roache, 1998), however, here $r_{21} \approx r_{32}$, and basic arithmetic can be used to calculate the order of convergence $\hat{p}$ (3.3) and the exact value for the variable of interest $f$ (3.4). $f_{exact}$ is the value of the variable of interest which would be obtained as $h_{cell} \to 0$.

$$h_{cell} = \left[ \frac{1}{N} \sum_{i=1}^{N} \Delta V_i \right]^{1/3}$$ (3.1)

$$r_{21} = \frac{h_{cell_2}}{h_{cell_1}} \approx r_{32} = \frac{h_{cell_3}}{h_{cell_2}}$$ (3.2)

$$\hat{p} = \frac{\ln |f_3 - f_2|}{\ln (r_{21})}$$ (3.3)

$$f_{exact} \approx f_1 + \frac{f_1 - f_2}{r_{21}^{\hat{p}} - 1}$$ (3.4)

Table 3.1 and Figure 3.1 display the efficiencies for each level of mesh refinement, and the extrapolated values. The percent differences between $\eta$ and its value at the previous level of refinement is also given to show the asymptotic reduction in changes to the variable of interest as the mesh is refined. The first refinement gives roughly a 4% increase in efficiency while the second only gives a 1% increase. The extrapolated values are only approximately half of a percent greater than the fine mesh results, and thus the fine mesh exhibits excellent convergence. Note that the refinement ratio is about 1.19 which is above the minimum value of 1.1 recommended to avoid error (Roache, 1998). It can also be seen that there is monotonic convergence for both scales and no oscillatory convergence. Although they are not shown, both $P^*$ and $\phi$ display monotonic/asymptotic convergence as well. Finally, the prototype $\eta$ is higher than the model $\eta$, which is expected because the prototype is at a much higher $Re$ than the model.
Table 3.1: Percent differences between efficiency at current level of mesh refinement and one level coarser, for single-phase flow.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>( h_{\text{cell}} )</th>
<th>( \eta_M )</th>
<th>% diff.</th>
<th>( \eta_P )</th>
<th>% diff</th>
</tr>
</thead>
<tbody>
<tr>
<td>N/A N/A</td>
<td>-</td>
<td>Coarse</td>
<td>0.055</td>
<td>0.770</td>
<td></td>
<td>0.782</td>
<td></td>
</tr>
<tr>
<td>N/A N/A</td>
<td>-</td>
<td>Medium</td>
<td>0.046</td>
<td>0.799</td>
<td>3.61</td>
<td>0.819</td>
<td>4.57</td>
</tr>
<tr>
<td>N/A N/A</td>
<td>-</td>
<td>Fine</td>
<td>0.039</td>
<td>0.808</td>
<td>1.15</td>
<td>0.828</td>
<td>1.02</td>
</tr>
<tr>
<td>N/A N/A</td>
<td>-</td>
<td>Extrap.</td>
<td>0.0</td>
<td>0.812</td>
<td>0.53</td>
<td>0.830</td>
<td>0.29</td>
</tr>
</tbody>
</table>

Figure 3.1: Convergence of efficiency with mesh refinement for single-phase flow.

3.2.2 Single-phase Results

Qualitative results of the single-phase flow through the fine mesh are presented and discussed in this section. Figure 3.2 shows the pressure contours of an axial slice taken right beneath the hub nose cone. All of the features expected of a close to BEP condition can be observed. For instance, the high pressure regions correspond well with the leading edges of the guide vane and runner blade. This implies the
flow is fairly incident on the leading edges where the stagnation point occurs. A very low pressure region is shown underneath the hub, where the cavitating vortex is expected to form. Low pressure regions also form on the leading edge suction-side of the blade which is another area prone to cavitation.

The hydrostatic pressure is shown in Figure 3.3. It is clearly evident that the gradient is much greater for the prototype. While this will not have any effect on the single-phase results, it will have a considerable effect on the differences between the model and prototype cavitating performance results, especially as performance breakdown occurs. Figure 3.4 shows the low static pressure (the static pressure is the sum of the piezometric pressure and the hydrostatic pressure) vortex core beneath the hub of the model and the prototype. The pressure recovery effect of the diffuser can be seen. Figure 3.5 displays the streamlines beneath the hub in the draft tube. The steady-periodic (although the draft tube is not periodic) RANS results clearly capture the hub vortex structure. The streamlines downstream of the elbow are fairly tangled (more so for the prototype) and this supports the WAH claim that this is a terribly performing draft tube.

Figure 3.6 shows the axial velocity of the flow in the model/prototype draft
Figure 3.3: Coordinate slice of draft tube displaying contours colored by hydrostatic pressure [Pa] for the single-phase simulations of the (a) model and (b) prototype.

Figure 3.4: Coordinate slice of draft tube displaying contours colored by static pressure [Pa] for the single-phase simulations of the (a) model and (b) prototype.

tubes. It is obvious, that there is a thin recirculating core beneath the hub nose cone. This is a problematic flow pattern, especially so close to BEP. The recirculation region beneath the hub vortex is well-known to form and breakdown the hub-vortex, and cause instabilities (Escudier, 1987; Leibovich, 1978; Lucca-Negro and O’doherty, 2001). In fact, flow injection at the hub nose cone has been shown to remove the recirculation region and thus the vortex rope (Foroutan and Yavuzkurt, Foroutan and Yavuzkurt; Susan-Resiga et al., 2010).
3.3 Cavitating Machine Performance Results

This section presents the machine performance results from the multiphase CFD simulations of the model and the prototype, compares them amongst each other, and compares them with the model experiments and the prototype results obtained by scaling the model experiments with standard methods. Before these results are presented, a mesh convergence study similar to the analysis done in Section 3.2.1, is conducted using both efficiency and the total volume of vapor in the computational domain as variables of interest for monitoring the mesh convergence.
of the multiphase flow. It is important to mention that, in all of the presented multiphase cases, the pressure of the vapor which forms from cavitation, does not deviate in a significant manner from the saturation pressure. For large-scale cavitation this is expected and well-documented (Franc and Michel, 2006).

### 3.3.1 Mesh Convergence Study

The methods of the mesh convergence study shown here for the multiphase flow are similar to that shown for single-phase flow in Section 3.2.1. The only difference is that, besides the machine efficiency, the total volume of vapor in the computational domain is a variable of interest as well. The volume of vapor is used as an extra parameter to ensure the convergence of the multiphase flow and the cavitation modeling. Case 4 ($\sigma = 0.52$) was chosen as the representative case to exhibit mesh convergence for a number of reasons. First, this is the lowest Thoma number case which could be compared to experimental results. Next, as $\sigma$ is steadily decreased, this is the first case which shows significant cavitation breakdown in the model, and it is cavitation breakdown which is of the most interest in this chapter. Finally, a large amount of vapor forms in the draft tube, runner, and guide vane regions, allowing for a quality test of the mesh convergence for multiphase flow.

As the mesh is refined, the efficiencies of the model and prototype with cavitation converge just as well as the single-phase cases, as evidenced in Table 3.2. There is little difference between the medium and fine meshes for both scales, and therefore the fine mesh results are close to the extrapolated values. Again, the prototype displays the expected higher efficiency than the model, that is caused by a much higher $Re$. It is also evident (Figure 3.7) that the CFD efficiencies, while underpredicted, are fairly close to the experimental/estimated results for both scales. Table 3.3 and Figure 3.8 provide the mesh convergence behavior of the percentage of vapor in the computational domain for both scales. This parameter converges at a slower rate than the efficiency. In fact, there is a large variance between the medium and coarse meshes. However, those variances decreases by a large amount between the medium and fine meshes, and display adequate convergence by the fine mesh. The model, which has less vapor than the prototype at this $\sigma$, accordingly converges at a faster rate than the prototype. This is indicative of the slightly faster efficiency convergence rate for the model. It should be noted
Table 3.2: Percent differences between efficiency at current level of mesh refinement and one level coarser, for multiphase flow.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>( h_{cell} )</th>
<th>( \eta_M )</th>
<th>% diff.</th>
<th>( \eta_P )</th>
<th>% diff</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>0.52</td>
<td>Coarse</td>
<td>0.055</td>
<td>0.764</td>
<td>0.786</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Medium</td>
<td>0.046</td>
<td>0.797</td>
<td>4.17</td>
<td>0.815</td>
<td>3.61</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>0.039</td>
<td>0.801</td>
<td>0.47</td>
<td>0.823</td>
<td>0.97</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Extrap.</td>
<td>0.0</td>
<td>0.802</td>
<td>0.11</td>
<td>0.826</td>
<td>0.35</td>
</tr>
</tbody>
</table>

that, because the model is simulated at the prototype geometric dimensions (with dynamic viscosity and gravity adjusted to account for scaling effects), the amount of vapor between scales can also be compared directly, rather than as a percentage of the total domain. All parameters of interest, along with \( P^* \) and \( \phi \) (not shown), converge monotonically and asymptotically. Based on this mesh convergence study, and the appreciable convergence of the percentage of vapor between medium and fine meshes, the fine mesh is utilized for the rest of the results in this chapter.

Table 3.3: Percent differences between percentage of vapor in computational domain at current level of mesh refinement and one level coarser.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>( h_{cell} )</th>
<th>( %vap )_M</th>
<th>% diff</th>
<th>( %vap )_P</th>
<th>% diff</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>0.52</td>
<td>Coarse</td>
<td>0.055</td>
<td>0.08</td>
<td>0.08</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Medium</td>
<td>0.046</td>
<td>0.10</td>
<td>20.8</td>
<td>0.12</td>
<td>32.0</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>0.039</td>
<td>0.11</td>
<td>9.01</td>
<td>0.14</td>
<td>14.1</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Extrap.</td>
<td>0.0</td>
<td>0.12</td>
<td>7.38</td>
<td>0.16</td>
<td>12.6</td>
</tr>
</tbody>
</table>
Figure 3.7: Convergence of efficiency with mesh refinement for multiphase flow and comparison with empirical/estimated results.
Figure 3.8: Convergence of vapor percentage in the computational domain with mesh refinement.
3.3.2 Multiphase Machine Performance Results

The multiphase machine performance results are presented here for the model and prototype scales. The values of the important non-dimensional performance flow variables such as specific energy coefficient $\psi$, flow coefficient $\phi$, power coefficient $P^*$, and efficiency $\eta$, are given in tables for each cavitation case (Cases 1-7). In those tables, the errors $\epsilon$ of the CFD flow variables, with respect to the empirical results, and with respect to the standard estimates based on the empirical results (in the prototype cases), are given. The subscripts are defined as ‘model experiment’ $\cdot \exp$, ‘model CFD’ $\cdot M_{CFD}$, ‘model’ $\cdot M$, ‘industry standard scaled prototype estimate’ $\cdot P_{sta}$, ‘prototype CFD’ $\cdot P_{CFD}$, and ‘prototype’ $\cdot P$. Along with each table is the corresponding plot of the data.

The net specific energy coefficients are shown in Table 3.4 and Figure 3.9. Recall that the model tests attempted to maintain a fairly constant $\psi$ as $\sigma$ was reduced. Even so, the experimental results still vary slightly. The penstock, partial-spiral casing, and stay vanes were not simulated in the periodic computations, but the losses through these components are assumed to have a negligible effect on the net specific energy, as shown by Vu and Retieb (2002) and Jošt and Lipej (2009) (especially since cavitation never formed in these regions in the experiments). Since a total pressure inlet condition, and a piezometric pressure outlet condition were used for the computations, the net specific energy was effectively set across the machine. The extremely small errors displayed here, prove that the BC is functioning properly and that the cases are being run at the correct operating conditions, including at the intended Thoma number.

It should be noted that the dynamic pressure (based on the slug-flow assumption) at the outlet is not set in the simulation, because the flow rate is converged upon by the computation. Thus, the total pressure inlet condition, to obtain the correct net specific energy, is determined by using the dynamic pressure (based on the slug-flow assumption) calculated from the experimental flow rate values. For Cases 5, 6, and 7, where no experimental flow rates are available, an estimate of the flow rate was made to calculate the inlet condition. It is possible to construct a more formal BC which would iteratively set the net specific energy as the solution converges. This was successfully accomplished by Panov et al. (2012). However, when using the slug-flow assumption, as is done with the empirical results, the outlet dynamic
Table 3.4: Net specific energy coefficients of all cases. CFD results are compared with empirical/estimated data to obtain errors.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>(\psi_{exp})</th>
<th>(\psi_{M,\text{CFD}})</th>
<th>(\epsilon_M)</th>
<th>(\psi_{P,\text{CFD}})</th>
<th>(\epsilon_P)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.24</td>
<td>Fine</td>
<td>0.7015</td>
<td>0.7035</td>
<td>0.28</td>
<td>0.7043</td>
<td>0.40</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>0.6990</td>
<td>0.7008</td>
<td>0.25</td>
<td>0.6998</td>
<td>0.11</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Fine</td>
<td>0.7045</td>
<td>0.7065</td>
<td>0.29</td>
<td>0.7079</td>
<td>0.49</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>0.6957</td>
<td>0.6969</td>
<td>0.18</td>
<td>0.6924</td>
<td>0.48</td>
</tr>
<tr>
<td>5</td>
<td>0.34</td>
<td>Fine</td>
<td>0.6966</td>
<td></td>
<td></td>
<td>0.6926</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.20</td>
<td>Fine</td>
<td>0.6980</td>
<td></td>
<td></td>
<td>0.6931</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.13</td>
<td>Fine</td>
<td>0.7045</td>
<td></td>
<td></td>
<td>0.6922</td>
<td></td>
</tr>
</tbody>
</table>

Pressure has little effect on the total pressure difference across the machine, and the estimates used here work well enough to provide net specific energy coefficients which hardly deviate (no more than 0.4% error) from the range of experimental values.

The flow coefficients \(\phi\) are displayed in Table 3.5 and Figure 3.10. The flow coefficients are within 1.9% of the experimental results. In fact, the model results are all within 1.0% of the experiments. The standard scaling of the model results to prototype scale assumes that the flow coefficient will remain the same. However, this assumption does not seem to have much physical basis, even for single-phase flow where lack of \(Fr\) similitude is not a problem. Since the prototype is at much higher \(Re\) than the model, it is intuitive that the flow coefficient through the prototype could be higher, as the boundary layers are thinner, resulting in less blockage effects. The CFD shows that the flow coefficients are in fact slightly larger for the prototype scale, and this occurs for the single-phase cases as well. This is one reason why it is important to use a total pressure inlet for prototype simulations, rather than a velocity inlet calculated from the model experiments, as a BC.

For Cases 1-4, the model and prototype flow rates display a similar trend with decreasing \(\sigma\), and this trend compares well with the experiments with the exception of Case 2. For Case 2, the experiments show a slight increase in flow rate as the small vaporous region on the leading-edge suction side of the runner begins to grow. This is a common phenomenon in slightly cavitating turbomachinery. Small amounts of vapor on the blades tends to boost the flow rate, power, and efficiency.
of the machine. This effect has been captured by CFD on a hydroturbine blade by Susan-Resiga et al. (2003) but is not seen in the CFD results of Panov et al. (2012). Here, for Case 2 of the model, even the fine mesh CFD had trouble capturing these small amounts of cavitation on the blade (capturing what seemed to be less vapor in this region than was shown on the experimental drawings), and the lack of capturing this vaporous region correctly, is expected to be the reason why the slight increases in flow coefficient, power coefficient, and efficiency are not seen in these results.

However, for Case 2, since the vapor does not affect the performance of the machine by a great deal, the results are still adequately close to the empirical data. Furthermore, as will be seen in the qualitative results presented shortly, the prototype experiences slightly more cavitation than the model in Cases 1-4, for each corresponding case. This effect is caused by the lack of $Re$ and $Fr$ similarity. For Case 3, both scales display a slight increase in $\phi$ and $P^*$, where enough vapor

Figure 3.9: Effect of Thoma number on net specific energy coefficient.
Table 3.5: Flow coefficients of all cases. CFD results are compared with empirical/estimated data to obtain errors.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>$\phi_{exp}$</th>
<th>$\phi_{M,CFD}$</th>
<th>$\epsilon_M$</th>
<th>$\phi_{P,CFD}$</th>
<th>$\epsilon_P$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.24</td>
<td>Fine</td>
<td>0.3993</td>
<td>0.3947</td>
<td>0.66</td>
<td>0.4032</td>
<td>0.97</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>0.4001</td>
<td>0.3933</td>
<td>0.98</td>
<td>0.4028</td>
<td>0.66</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Fine</td>
<td>0.3973</td>
<td>0.3956</td>
<td>0.25</td>
<td>0.4048</td>
<td>1.89</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>0.3882</td>
<td>0.3868</td>
<td>0.19</td>
<td>0.3952</td>
<td>1.82</td>
</tr>
<tr>
<td>5</td>
<td>0.34</td>
<td>Fine</td>
<td></td>
<td>0.3774</td>
<td></td>
<td>0.3867</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.20</td>
<td>Fine</td>
<td></td>
<td>0.3587</td>
<td></td>
<td>0.3731</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.13</td>
<td>Fine</td>
<td></td>
<td>0.3477</td>
<td></td>
<td>0.3658</td>
<td></td>
</tr>
</tbody>
</table>

is seemingly present to boost these parameters, and too little to begin to cause performance degradation. The lack of $Fr$ similarity between the two scales becomes more evident as $\sigma$ decreases further, as in Cases 5, 6, and 7. Much more cavitation forms here in the model cases than in the corresponding prototype cases, especially in the draft tube, and the model flow coefficient begins to drop off more sharply than the prototype. This effect is even more pronounced when observing the power and efficiency breakdown. Physically, as cavitation breakdown begins to occur, the flow rates decrease because the relatively low-density vapor induces blockage and recirculation regions which grow with vapor volume. Therefore, the fact that the flow coefficients, power coefficients, and efficiencies, differ between model and prototype with little to no cavitation is predominantly due to lack of $Re$ similarity, while the fact that this difference increases with decreases in Thoma number is primarily caused by lack of $Fr$ similarity.

Table 3.6 and Figure 3.11 show the power coefficients for varying Thoma number. The model CFD results are within $2.7 - 3.7\%$ of the model experiments in Cases 1, 3, and 4. As with the flow coefficient, and expected to be caused by the same conclusion reached with $\phi$, Case 2 does not display the boost in $P^*$ due to small amounts of leading-edge suction-side runner blade cavitation. The prototype CFD power coefficient is higher than the model CFD for all cases, and this is expected due to higher $Re$ in the prototype. While both CFD scales slightly under-predict $P^*$, the increase in $P^*$ between the model CFD and prototype CFD, for any given Case 1-4, is quite similar to the increase between the model experiment and the prototype standard results for that case. Thus, the prototype CFD results are
Figure 3.10: Effect of Thoma number on flow coefficient.

within 2.5 – 3.9% of the prototype standard results for Cases 1, 3, and 4. Again, Case 2 shows slightly more error due to the aforementioned reasons. As with $\phi$, Case 3 of both CFD scales show a power increase because not enough vapor has yet to form to breakdown the torque. Cases 5, 6, and 7 begin to display a fairly sharp breakdown in $P^*$, and the breakdown is more pronounced for the model where the amount of cavitation is much more severe for these cases. Now, the difference between the model and the prototype clearly deviates from what is predicted based on the single-phase flow standard scaling. The power breakdown occurs because the torque on the runner is decreased ($\Omega$ is constant) by two major, and intimately coupled, factors. The first is that the flow rate through the entire machine decreases, for reasons previously discussed. The second, and more localized reason, is that as the vapor region on the suction-side of the blade increases with decreasing $\sigma$, the net pressure difference across the blade is decreased. The vapor region remains close to the vapor pressure,
Table 3.6: Power coefficients of all cases. CFD results are compared with empirical/estimated data to obtain errors.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>$P_{M,exp}$</th>
<th>$P_{M,CFD}$</th>
<th>$\epsilon_M$</th>
<th>$P_{P,sta}$</th>
<th>$P_{P,CFD}$</th>
<th>$\epsilon_P$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.24</td>
<td>Fine</td>
<td>0.2337</td>
<td>0.2252</td>
<td>3.65</td>
<td>0.2449</td>
<td>0.2352</td>
<td>3.94</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>0.2353</td>
<td>0.2226</td>
<td>5.40</td>
<td>0.2464</td>
<td>0.2346</td>
<td>4.77</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Fine</td>
<td>0.2335</td>
<td>0.2262</td>
<td>3.12</td>
<td>0.2446</td>
<td>0.2385</td>
<td>2.48</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>0.2219</td>
<td>0.2159</td>
<td>2.72</td>
<td>0.2326</td>
<td>0.2253</td>
<td>3.16</td>
</tr>
<tr>
<td>5</td>
<td>0.34</td>
<td>Fine</td>
<td>0.2050</td>
<td></td>
<td></td>
<td>0.2163</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.20</td>
<td>Fine</td>
<td>0.1754</td>
<td></td>
<td></td>
<td>0.1959</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.13</td>
<td>Fine</td>
<td>0.1573</td>
<td></td>
<td></td>
<td>0.1818</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

and therefore as the vapor region grows with decreasing Thoma number, a larger region of the blade is restricted from maintaining the larger pressure difference it had with less vapor.

Finally, the efficiencies are displayed in Table 3.7 and Figure 3.12. The model CFD efficiencies of Cases 1, 3, and 4 are within 3% of the model experiments, while Case 2, again not predicting the slight increases in power and flow rate is under-predicted by 4%. As cavitation breakdown begins, between Cases 3 and 4, the trend of the efficiency of the model and the prototype matches well with the experiments.

The prototype CFD efficiency does not show the expected increase relative to the model CFD efficiency from running at higher $Re$, and this results in slightly higher error than in the model CFD. This occurs in the single-phase results as well. Furthermore, Weir American Hydro (WAH) mentioned that the field testing was done on this particular runner, and the full-scale machine, while having better performance than the model, did not reach the expected performance either (Joe Hill of Weir American Hydro, personal communication, December 2013). Investigating Case 1, where the least cavitation occurs and the standard scaling of efficiency is expected to be more accurate, the problem can be uncovered. The standard method predicts an efficiency increase of 0.036 between model and prototype. The CFD only shows an increase of 0.017 for Case 1. The increase in the power coefficient between model and prototype CFD matches well with what is predicted by the standard scaling. However, the flow coefficient was expected to be the same at both scales, while the CFD shows it is higher for the prototype. Since the flow rate
is in the denominator of the efficiency, the higher prototype CFD flow coefficient may result in the lower than expected efficiency seen in the prototype CFD. To demonstrate this, if the model CFD flow rate is placed in the calculation of the prototype CFD efficiency, thereby assuming the flow rate does not change between model and prototype, the difference between model CFD $\eta$ and prototype CFD $\eta$ becomes 0.035, which is much closer to 0.036. This seems to imply, at the very least, that the lower than expected prototype CFD efficiencies are due to the unaccounted for increase in flow rate at higher $Re$.

Cases 5, 6, and 7 show the severe breakdown in efficiency for both the prototype and the model. By Case 7, the model efficiency has decreased to 0.64 and the prototype efficiency to 0.72. Thus, while both scales show substantial reductions in $\eta$, the model reductions are much sharper. Table 3.8 displays the percent difference between the model CFD and prototype CFD efficiencies, as well as the percent difference between the model experiments and standard scaled prototype results.
Table 3.7: Efficiency of all cases. CFD results are compared with empirical/estimated data to obtain errors.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>η_{M,exp}</th>
<th>η_{M,CFD}</th>
<th>ε_{M}</th>
<th>η_{P,sta}</th>
<th>η_{P,CFD}</th>
<th>ε_{P}</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.24</td>
<td>Fine</td>
<td>0.8342</td>
<td>0.8111</td>
<td>2.77</td>
<td>0.8724</td>
<td>0.8282</td>
<td>5.07</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>0.8411</td>
<td>0.8077</td>
<td>3.98</td>
<td>0.8793</td>
<td>0.8325</td>
<td>5.32</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Fine</td>
<td>0.8339</td>
<td>0.8092</td>
<td>2.97</td>
<td>0.8720</td>
<td>0.8321</td>
<td>4.58</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>0.8217</td>
<td>0.8007</td>
<td>2.55</td>
<td>0.8597</td>
<td>0.8231</td>
<td>4.25</td>
</tr>
<tr>
<td>5</td>
<td>0.34</td>
<td>Fine</td>
<td></td>
<td>0.7798</td>
<td></td>
<td></td>
<td>0.8073</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.20</td>
<td>Fine</td>
<td></td>
<td>0.7003</td>
<td></td>
<td></td>
<td>0.7574</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.13</td>
<td>Fine</td>
<td></td>
<td>0.6422</td>
<td></td>
<td></td>
<td>0.7180</td>
<td></td>
</tr>
</tbody>
</table>

Table 3.8: Percent differences between model/prototype scales for CFD and empirical/estimate results.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>100(η_{P,sta}−η_{M,exp})</th>
<th>η_{P,sta}</th>
<th>100(η_{P,CFD}−η_{M,CFD})</th>
<th>η_{P,CFD}</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.24</td>
<td>Fine</td>
<td>4.4</td>
<td>4.4</td>
<td>2.1</td>
<td>2.1</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>4.4</td>
<td>4.4</td>
<td>3.0</td>
<td>3.0</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Fine</td>
<td>4.4</td>
<td>4.4</td>
<td>2.8</td>
<td>2.8</td>
</tr>
<tr>
<td>4</td>
<td>0.52</td>
<td>Fine</td>
<td>4.4</td>
<td>4.4</td>
<td>2.7</td>
<td>2.7</td>
</tr>
<tr>
<td>5</td>
<td>0.34</td>
<td>Fine</td>
<td>3.4</td>
<td>3.4</td>
<td>3.4</td>
<td>3.4</td>
</tr>
<tr>
<td>6</td>
<td>0.20</td>
<td>Fine</td>
<td>7.5</td>
<td>7.5</td>
<td>7.5</td>
<td>7.5</td>
</tr>
<tr>
<td>7</td>
<td>0.13</td>
<td>Fine</td>
<td>10.6</td>
<td>10.6</td>
<td>10.6</td>
<td>10.6</td>
</tr>
</tbody>
</table>

For Case 7, the model efficiency is almost 11% less than the prototype efficiency at the same Thoma number. Again, this is predominantly caused by the lack of $Fr$ similarity between the scales. The physical reasons for the efficiency breakdown were discussed when the results for $\phi$ and $P^*$ were presented.
Images of the model cavitation for Case 1 ($\sigma = 1.24$) are shown in Figure 3.13. This fine mesh image shows that cavitation is only captured beneath the guide vane overhang. The mixing plane downstream of the guide vane prevents the vapor from transporting into the runner region because this small amount of vapor is mostly circumferentially averaged out. However, the vapor beneath the guide vane is detached and in reality could easily be transported downstream and impact the front of the runner blade, possibly causing more severe rotor-stator interactions than would occur in single-phase flow. This is something that would only be captured by the full unsteady computations. Cavitation occurs at the guide vane overhang in every case, for the model and prototype, to approximately the same extent. It does not seem to have a large effect on the steady-state performance results and will not be discussed further for the periodic cases.

Figure 3.14 shows the experimental photos and drawings of cavitation. The guide vane cavitation, which was captured in the computations, was noted in the
experiments as well. A very small amount of cavitation is noted on the suction-side of the runner, by the shroud. The pressures from the computation are just shy of the saturation pressure in this region, so they do not display vapor. This is also true for the hub vortex region. The thin precessing rope cavitation is thought to occur within the recirculating region of the hub vortex, where the lowest pressures occur and are within a few thousand Pascals of the vapor pressure. There are several reasons as to why the vapor may not be captured on the leading-edge suction-side of the runner and the thin hub vortex core, in this slightly cavitating computation. The first is that the mesh may have to be finer or of better quality. Since the pressures are close to saturation pressure in these regions, it may just be a matter of resolution, or possibly cell skewness. As these small amounts of vapor do not considerably affect the steady-state performance, the machine efficiency still compares well with the experiments after the presented mesh convergence study. The second is that unsteady simulations or improved turbulence modeling may be necessary to capture the lower pressures, and thus cavitation in these regions. Finally, the formation of these small-scale cavitation features is very susceptible to the initial/boundary conditions. Since this is a periodic computation, beginning at the guide vane inlet, flow conditions such as the inlet swirl angle needed to be estimated. This estimation may have an effect on whether or not vapor forms in those regions. This can only be overcome with simulations of the entire machine, where the swirl entering the guide vanes is captured correctly from the flow in the penstock, partial-spiral casing, and stay vanes.

Figure 3.15 shows the fine mesh model results for Case 2 ($\sigma = 0.99$). Vapor forms beneath the guide vane and also beneath the hub. Both of these features were seen in the experiments (Figure 3.16). The CFD hub cavitation displays what the experimenters documented as ‘torch-like’ directly beneath the nose cone, and a thin cavitating (but cloudy) vortex core structure farther downstream. However, the CFD does not seem to capture the full length of the thin cavitating vortex core as it does not convect as far into the draft tube as the photos. Again, either there is not enough resolution to capture that behavior, more effective turbulence modeling and unsteady computations are needed, or the components before the guide vanes need to be simulated. Although not pictured because the volume fraction is less than 10%, a small amount of vapor is captured on the leading-edge suction-side of the blade at the shroud, in a similar location as in the experimental diagrams. The
Figure 3.13: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 1 ($\sigma = 1.24$).

Figure 3.14: Cavitation drawings and photo from Case 1 ($\sigma = 1.24$) model tests.

amount of vapor seems to be less than what is pictured in the diagrams, and as previously discussed, this may be the reason why $\phi$, $P^*$, and $\eta$ do not show the same slight increases in the CFD as they do in the experiments.

Figure 3.17 displays the fine mesh model results for Case 3 ($\sigma = 0.70$). Vapor is clearly visible beneath the guide vane, on the shroud, on the leading-edge suction-side of the runner blade, and beneath the hub. These features were also documented in the experiments (3.18). The experiments mention vapor forming on the shroud and leading-edge suction-side of the blade and being transported downstream within the blade passage. This formation and transport was captured with the CFD. The
Figure 3.15: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 2 ($\sigma = 0.99$).

Figure 3.16: Cavitation drawings and photo from Case 2 ($\sigma = 0.99$) model tests.

experiments show a large torch-like structure and the thin-core vortex is not visible anymore. The computations capture the large structure as it now extends into the draft tube. The vapor is fairly axisymmetric near the hub but still shows evidence of spiral breakdown as it enters the sharp expansion of the draft tube.

Figure 3.19 displays the fine mesh model results for Case 4 ($\sigma = 0.52$). All of the features visible in Case 3 have now become more exaggerated. Furthermore, it is now evident that the vapor on the suction-side of the runner blades, detaches from the leading edge and travels over the blade. This is thought to be cloud cavitation, which has a high propensity to cause damage on the surface of the blade and to
Figure 3.17: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 3 ($\sigma = 0.70$).

Figure 3.18: Cavitation drawings and photo from Case 3 ($\sigma = 0.70$) model tests.

the shroud. Further investigation is also necessary to determine if this structure is an interblade vortex, because it seems to occur in a similar region to this flow feature. A reentrant jet forms along the shroud and is visible as a ripple in $\alpha_v$ here. Hence, the flow is also beginning to detach from the shroud. The detached cavitation from the leading-edge now extends past the trailing-edge of the runner blade and to the mixing plane at the inlet to the draft tube. The flow beneath the hub shows much more cavitation and a longer axisymmetric torch structure. The torch eventually becomes a spiral type vortex breakdown as it interacts with a large region of recirculating flow, and the vapor wraps around this recirculation region.
Figure 3.19: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 4 ($\sigma = 0.52$).

These results compare qualitatively well to the experimental drawings, photos, and comments (Figure 3.20).

Figure 3.20: Cavitation drawings and photo from Case 4 ($\sigma = 0.52$) model tests.

Figure 3.21 shows the fine mesh model results for Case 5 ($\sigma = 0.34$). The shroud is almost completely covered by vapor, and this vapor clearly detaches as an even larger recirculation region forms than the previous cases. Besides the shroud, the vapor in the runner passage remains entirely on the suction-side of the blade, and the detached vapor transported through the passage has grown wider in the span-wise direction. The vapor transported through the runner passage is still
Figure 3.21: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 5 ($\sigma = 0.34$).

not extensive enough to pass through the circumferential averaging of the mixing plane and into the draft tube for $\alpha_v = 0.2$, but is present in the draft tube at lower vapor volume fractions. The hub torch cavitation has become wider and completely axisymmetric beneath the nose cone (and only fails to remain so as it reaches the draft tube elbow). Furthermore, cavitation has formed on the walls of the draft tube at the elbow where the flow is sharply turned towards the outlet. All of these features are consistent with the severe performance breakdown with occurs between Cases 4 and 5.

Figure 3.22 displays the fine mesh model results for Case 6 ($\sigma = 0.20$). While the vapor formation still remains only on the suction-side of the blade (besides the formation on the entire shroud surface area), it is now extensive enough to pass through the circumferential averaging of the mixing plane at $\alpha_v = 0.2$. This is visualized as two isosurfaces which are transported through the draft tube. They are at a greater radius than the hub torch but smaller radius than the draft tube walls. The hub torch has become even wider, remains as axisymmetric as Case 5, and extends even farther into the draft tube. The cavitation at the start of the draft tube elbow, seen in Case 5, now covers the entire elbow wall in Case 6.
Figure 3.22: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 6 ($\sigma = 0.20$).

Figure 3.23 provides the fine mesh model results for Case 7 ($\sigma = 0.13$). The most apparent difference between Case 6 and 7 is that in Case 7 the vapor detaching from the runner blades is now transported much farther into the draft tube. Besides that feature, Case 7 displays the most hub cavitation of any case, and it is very axisymmetric. Furthermore, the vapor on the draft tube elbow wall has become even more substantial. However, the vapor in the runner region does not vary much between Cases 6 and 7. Moreover, it is visually clear that most of the new vapor seen in Cases 6 and 7, that wasn’t seen in Case 5, is in the draft tube region. This suggests that the difference in the losses between Cases 5, 6, and 7, causing severe efficiency breakdown, occurs primarily in the draft tube. This is a statement which will be quantitatively verified in Section 3.4.

The prototype CFD results for the fine mesh can be seen in Figures 3.24, 3.25, 3.26, 3.27, 3.28, 3.29, and 3.30. The prototype results display similar cavitation patterns to the corresponding model cases. However, cavitation in the prototype occurs to a slightly greater extent than in the model, in Cases 1-4, and to a substantially lesser degree in the prototype than in the model, in Cases 5-7. The contrast of vapor content between the model and prototype seen in Cases 1-4 is
caused by lack of both \( Re \) and \( Fr \) scaling, while the contrast seen in Cases 5-7 is predominately due to lack of \( Fr \) scaling. One feature of the prototype which is clearly different from the model is its ability to avoid extreme cavitation in the draft tube. When the runner/hub cavitation extends to lower elevation (into the draft tube), the higher hydrostatic pressure gradient quickly thins out the vapor, where with the model, the vapor is transported farther downstream. Another feature is that when the vapor on the shroud begins to recirculate, it does so at a lower elevation than the model. Furthermore, besides obstructing the vapor transport into the draft tube, the prototype prevents/limits the formation of vapor that occurs on the draft tube walls in Cases 6 and 7 of the model. Again, this is attributed to the higher static pressures in the prototype draft tube, caused by gravitational effects. Finally, the amount of cavitation occurring in the runner region is fairly similar between the corresponding model and the prototype cases, especially at lower Thoma numbers. This seems to suggest that the losses in the prototype and model runners, at lower Thoma numbers, are similar, and that the differences in draft tube vapor (due to lack of \( Fr \) similitude) may be the cause of the more pronounced performance breakdown in the model. These effects will
be quantified in Section 3.4 to show the differences that the lack of $Re$ and $Fr$ similitude have on the losses in each stage of the model and prototype machines.

Figure 3.24: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 1 ($\sigma = 1.24$).
Figure 3.25: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 2 ($\sigma = 0.99$).

Figure 3.26: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 3 ($\sigma = 0.70$).
Figure 3.27: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 4 ($\sigma = 0.52$).

Figure 3.28: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 5 ($\sigma = 0.34$).
Figure 3.29: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 6 ($\sigma = 0.20$).

Figure 3.30: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for Case 7 ($\sigma = 0.13$).
3.4 Stage-by-Stage Performance Decomposition

The three stages of the steady-periodic computational results (guide vanes, runner, and draft tube) are now decomposed to study the performance of the individual machine components. Similar stage-by-stage performance decompositions have been published with single-phase CFD results for the entire operating range of a Francis turbine (Vu and Retieb, 2002), and for various flow rates of a Francis turbine (Jošt and Lipej, 2009). However, this is the first time it is presented for cavitating flow in a hydroturbine. Here, the parameter of the study which is directly varied is not \( \psi \) nor \( \phi \), but Thoma number \( \sigma \).

The critical quantities which must be calculated for this analysis are the flux of potential energy \( \Pi \) (3.5), and the flux of kinetic energy \( K \) (3.6), through the inlet and outlet surfaces of each stage. The units of \( \Pi \) and \( K \) are Watts [W] and the inlet and outlet surfaces of each stage are shown in Figure 3.31.

\[
\Pi \equiv \int_S p(v \cdot n) dS \quad (3.5)
\]

\[
K \equiv \int_S \rho (v \cdot n)^2 (v \cdot n) dS \quad (3.6)
\]

The total flux of mechanical energy \( M \) can then be obtained with

\[
M \equiv \Pi + K. \quad (3.7)
\]

In previous calculations of machine efficiency (\( \eta \)), \( v \) used at the draft tube exit in the calculation of \( M \), was based on the slug-flow assumption (\( ||v|| = \frac{Q}{S_{out}} \)). This was done to compare to the experimental \( \eta \) which was calculated in the same manner. If the flow exits the draft tube completely normal to the exit plane, then this assumption is perfectly accurate. If it does not in some locations, or if there is reverse flow, then there are errors inherent in the assumption. The experiments did not have the luxury of obtaining detailed velocity fields at the draft tube exit (nor at any other locations throughout the machine). However, the CFD does provide this information, and the integrated quantities of \( \Pi \) and \( K \) can be obtained by using the complete velocity field \( v \) at some surface of interest. Since the slug flow assumption will not provide the most accurate results in the stage-by-stage
decomposition, the complete velocity field is used to calculate $M$ in this section. Recalculating the machine efficiency in this manner, and referring to this as the true machine efficiency ($\eta_{true}$), overall higher values of machine efficiency are obtained and shown in Figure 3.32.

$\eta_{true}$ shows similar model/prototype trends as $\eta$, with lower model efficiency which diverges from the prototype efficiency as $\sigma$ decreases. The percent difference of $\eta_{true}$ between the scales is 1.6% for Case 1, and increases to over 7% by Case 7. Case 5 ($\sigma = 0.34$) shows a jump in model efficiency which is not obtained when using the slug-flow assumption at the draft tube outlet. In this case the model and prototype true machine efficiencies are practically the same. This unexpected result will appear in the stage-by-stage decomposition as well, and will be discussed more thoroughly then. Here, it is pointed out to show that this effect occurs in the analysis presented in this section, and not in the previous results, because of the choice of how the velocity at the draft tube exit is calculated.

To calculate the efficiency of the runner, the general efficiency equation, defined as the power transferred to the runner by the fluid, over the change in the flux of mechanical energy of the fluid between the inlet and outlet of the runner stage, is
Figure 3.32: Effect of Thoma number on true machine efficiency.

\[ \eta_{\text{run}} = \frac{P}{\Delta M_{\text{run}}} \]  

\( \Delta \) represents the difference of a quantity between the inlet and outlet surfaces. Figure 3.33 displays the runner-only efficiencies of the model and the prototype. Since \( \Delta M_{\text{run}} \) only accounts for flow through the runner stage, there are much less losses than through the entire machine, and the runner-only efficiencies can be over 95% at BEP. With no cavitation in the runner passage, \( \eta_{\text{run}} \) is close to 96% for the model, and over 97% for the higher \( Re \) prototype. As cavitation begins to form, the model runner efficiency initially shows little variation until it begins to decrease slightly in Case 4. This corresponds with the extension of the vapor on the suction-sides of the blades to the runner outlet, which was the condition mentioned by Avellan (2004) as when the machine efficiency drop could be first noticed in a cavitation test. As \( \sigma \) is lowered further the model runner efficiency is reduced more sharply between Cases 4 and 6, by almost 0.04. Between Cases 6
Figure 3.33: Effect of Thoma number on runner-only efficiency.

and 7, the model runner efficiency levels off and even increases a slight amount. The prototype runner efficiency displays a decrease as cavitation begins to form on the blades in Case 3. This is before the vapor extends to the runner exit. The prototype $\eta_{run}$ gradually decreases as $\sigma$ is reduced, but at a slightly greater rate than the model, so that for the lowest Thoma numbers, the prototype $\eta_{run}$ seems to approach the model $\eta_{run}$. This is consistent with the images showing that the vapor on the model and prototype runner look similar at the lower $\sigma$ values. The fact that this occurs may be due to the reference elevation chosen for the model tests. Roughly located at the mid-elevation-point of the guide vanes, the runner blade does not extend very far below it (about 2 meters). Thus, the lack of $Fr$ similarity at prototype-scale is less of an issue at these elevations. As $\sigma$ is decreased below 0.20, the prototype runner efficiency show signs of leveling off as well, at about the same value as the model.

With the runner-only efficiencies of the model and prototype approaching each
other for the most extensive cavitation cases, the question as to which stage causes
the separation between model/prototype machine efficiency at low Thoma numbers
now has an obvious answer. While the losses in the guide vanes are expected
to be minimal for single-phase flow (Vu and Retieb, 2002), and relatively small
amounts of vapor formation occur in this region in the cavitating cases, along
with elevations here being near the reference, it is evident that the draft tube
must be the primary culprit. This can be shown quantitatively, with draft tube
analysis methods employed for single-phase flow by Susan-Resiga et al. (2010). A
dimensionless loss coefficient can be defined as
\[
\zeta \equiv \frac{M_{in} - M_{out}}{K_{in}} = \left(1 - \frac{K_{out}}{K_{in}}\right) - \frac{\Pi_{out} - \Pi_{in}}{K_{in}}
\]  
(3.9)
which is composed of the kinetic energy recovery coefficient
\[
C_{KR} \equiv 1 - \frac{K_{out}}{K_{in}}
\]  
(3.10)
and the potential energy recovery coefficient
\[
C_{PR} \equiv \frac{\Pi_{out} - \Pi_{in}}{K_{in}}.
\]  
(3.11)
An inequality that \(C_{KR}\) and \(C_{PR}\) must satisfy was defined by Susan-Resiga et al.
(2010), following the subsequent line of reasoning. For viscous flow, the loss
coefficient must be positive, and thus, \(C_{KR} > C_{PR}\) to satisfy (3.9). Since the draft
tube is a diffuser and the flow is decelerated between the inlet and the outlet,
\(K_{out} < K_{in}\), and therefore \(C_{KR} < 1\). Additionally, a diffuser usually results in a
pressure rise between inlet and outlet, \(\Pi_{out} > \Pi_{in}\), and \(C_{PR} > 0\). The conclusion
that was reached is the following inequality
\[
0 < C_{PR} < C_{KR} < 1.
\]  
(3.12)
Finally, a quantity called the kinetic-to-potential energy conversion ratio, which is
commonly used to quantify the performance of a draft tube, can be defined as
\[
\chi \equiv \frac{\Pi_{out} - \Pi_{in}}{K_{in} - K_{out}} < 1.
\]  
(3.13)
Figure 3.34: Effect of Thoma number on kinetic energy recovery coefficient.

Figure 3.34 shows that, with hardly any cavitation (and similarly for the single-phase case not shown here), the kinetic energy recovery coefficient is approximately the same for the model and prototype. As Thoma number is decreased, $C_{KR}$ of the model and prototype follow the same trend, although the model drops off more sharply than the prototype after Case 4. The values of $C_{KR}$ for the flows without much cavitation, about 0.72, are slightly less than values reported by Susan-Resiga et al. (2010) for single-phase flow of an axisymmetric draft tube at BEP (which were about 0.78).

The values of $C_{PR}$, shown in Figure 3.35, for the flows without much cavitation are between 0.5 and 0.6, which is a great deal less than the values of Susan-Resiga et al. (2010) at BEP. In fact, the $C_{PR}$ of Susan-Resiga et al. (2010) does not reach such low values until at 91 percent of BEP. Thus, even when cavitation is not a performance issue, this draft tube is incredibly poor at recovering the static pressure. For Cases 3 and 4, the model $C_{PR}$ decreases slightly as more cavitation
forms in the draft tube. However, for Case 5, as even more cavitation forms in the machine, and the axi-symmetric torch-like structure beneath the hub becomes more apparent, $C_{PR}$ returns to values close to the cases with hardly any cavitation at all. It is thought that this rise in $C_{PR}$ between Cases 4 and 5, may have to do with the transition of the flow beneath the hub from the thin vortex core to the axisymmetric torch structure, or the initial formation of cavitation on the draft tube walls, although this behavior could easily be caused by effects taking place upstream of the draft tube as well. In Cases 6 and 7, as the axisymmetric torch structure fully forms, cavitation begins to form on the draft tube elbow wall, and vapor is transported from the runner region into the draft tube, $C_{PR}$ drops off sharply. As Thoma number is decreased in the prototype, the prototype $C_{PR}$ shows a trend of gradual increase before it begins to drop off in cases 5, 6, and 7 (but not nearly as abruptly as the model).

To quantify the meanings of Figures 3.34 and 3.35, it is easier to observe the
loss coefficient $\zeta$, which is just the difference between the two recovery coefficients, in Figure 3.36. The loss coefficient in the model draft tube increases by 64\% between the highest and lowest Thoma numbers, which is comparable to the 67\% increase in the single-phase axisymmetric results of Susan-Resiga et al. (2010) between BEP and 91 percent of BEP. The prototype only displays a 33\% increase in loss coefficient between the highest and lowest Thoma numbers, and even shows performance improvements before the losses begin to increase in Cases 5, 6, and 7. The cases with the least cavitation (and the single-phase case not pictured) display a loss coefficient which is slightly higher in the prototype than the model. This is counter-intuitive because it is expected that the higher $Re$ flow that is not greatly affected by cavitation, would lead to less losses, and not more. However, Figure 3.5 shows that the streamlines of the prototype, downstream the draft tube elbow, are much more tangled than the model streamlines for single-phase flow leading to a visual explanation of the results seen in Figure 3.36. Quantitatively, since $C_{KR}$ is about the same in these cases for the model and prototype, it is $C_{PR}$ which causes this phenomenon. Ultimately, in these cases, the losses are greater in the prototype because the ratio of the potential energy to kinetic energy entering the draft tube is lower than in the model. Finally, it should be noted that the loss coefficients are an order of magnitude higher than those presented in Susan-Resiga et al. (2010), attesting to the poor performance of the draft tube under investigation in this dissertation.

The kinetic-to-potential energy conversion ratio can be seen in Figure 3.37, which along with Figure 3.36 gives the complete picture of the hydraulic performance of the draft tube. $\chi$ is simply the ratio of $C_{PR}$ to $C_{KR}$, and is commonly referred to as a measure of draft tube efficiency. Figure 3.37 shows that $\chi$ follows the same trends which have been discussed for $C_{PR}$. Susan-Resiga et al. (2010) report a $\chi$ of about 0.97 at BEP, and a decrease in $\chi$ of 7\% between BEP flow rate and 91\% BEP flow rate (9\% decrease in flow rate), for their single-phase axisymmetric draft tube. For the draft tube under investigation here, the single-phase and slightly cavitating cases (Cases 1 and 2), have $\chi$ values of approximately 0.8 and 0.7, for the model and prototype, respectively. This again attests to the poor performance of this draft tube. Between Cases 1 and 7, the model displays a reduction in $\chi$ of roughly 53\% while the prototype only displays a reduction of 24\%. Between Cases 1 and 7, the model flow rate decreases by 11.9\% and the prototype flow
rate by 9.3%. It is also evident that the draft tube has worse efficiency at higher Thoma number for the prototype than for the model. This is the same trend that was seen with $\zeta$ and was already discussed. Again, this is physically un-intuitive because the prototype is at higher $Re$. It is not apparent what the physical source of this phenomena is, and perhaps further investigation is necessary. Regardless, as Thoma number is decreased, the performance of the model and prototype draft tube diverge, with the model draft tube performing much more poorly at lower $\sigma$. Considering this, and that the model and prototype runner efficiencies approach each other with decreasing Thoma number, it is obvious that as $\sigma$ is decreased, the divergence of model and prototype machine efficiencies are due to the draft tube. However, this result should not be overstated, as vapor passage through the mixing planes increases with decreasing $\sigma$. This can cause unphysical vapor content to appear, and the results of this performance analysis at low $\sigma$ may be related to the limits of the mixing plane approach.
The losses in mechanical energy flux through each stage can also be examined, compared with one another, and used to inspect stage-by-stage performance differences between the model and prototype scales. A new variable $\Delta \Xi$ is defined, that is equivalent to the power lost between the inlet and outlet of a stage, which is not used to do work.

$$\Delta \Xi \equiv \Delta M - P = \Delta M - T\Omega$$  \hspace{1cm} (3.14)

In the guide vane and draft tube stages, since $\Omega = 0$, $\Delta \Xi$ is simply the change in the flux of mechanical energy between the inlet and outlet. In the runner, since work is done to rotate the turbine at non-zero $\Omega$, then $T\Omega$ must be subtracted away from the change in mechanical energy flux in the stage to determine $\Delta \Xi$. Letting $\Delta \Xi_{\text{stage}}$ and $\Delta \Xi_{\text{total}}$ represent the power losses not used to do useful work across a stage and the entire machine, respectively, the percentage of the losses in each
stage (for model and prototype), can be analyzed. Furthermore, these percentages, as well as the dimensional values of $\Delta \Xi$ can be directly compared across model and prototype scales. This can only be done because of how the CFD simulations are conducted. Recall that the prototype-scale geometry is used for the model-scale simulations, and that viscosity $\mu$ and gravity $g$ are altered to account for lack of $Re$ and $Fr$ scaling. The power coefficient, defined as $P^* = \frac{2P}{\pi \rho l \Omega^3 R^5}$, will only change in the CFD results as power $P$ itself changes, because $\rho$, $\Omega$, and $R$ are the same values for the model and prototype simulations. Thus, comparing dimensional power variables across scales is acceptable in these cases.

Figure 3.38 displays the percentage of the losses across the model guide vanes, runner, and draft tube. At the slightly cavitating conditions the draft tube accounts for over 40% of the losses, while the runner and guide vanes make up approximately 30% each. As vapor increases in the machine, the guide vane losses decrease fairly monotonically to about 10% for the lowest Thoma number. Cases 5-7, the three lowest Thoma numbers, show the losses in the draft tube increasing to almost 70% of the total, while the runner decreases its losses to approximately 20%. This helps quantify the effect that cavitation has on the various components of the model, by showing that the model draft tube develops the most severe losses as more vapor forms in the machine. Case 5 shows the decreased losses in the draft tube that were displayed in Figures 3.36 and 3.37, alongside an increased percentage of total losses occurring in the other components for that case.

Figure 3.39 displays the percentage of the losses across the prototype guide vanes, runner, and draft tube. At the slightly cavitating conditions the draft tube accounts for about 70% of the losses in the prototype, which is a much higher percentage than in the model. The prototype runner and guide vanes only account for approximately 20% and 10% of the losses, respectively, for the slightly cavitating cases. These results are consistent with data already shown, that displays lower draft tube efficiency, yet higher runner and machine efficiency for the prototype-scale, when compared to the model-scale. Surprisingly, as more cavitation forms the percentage of draft tube losses decreases to almost 40% before rising again to above 60% for the lowest Thoma number. On the other hand, the percentage of runner losses increases to a slightly higher value than the draft tube in Case 5, before it drops again when the draft tube losses increase in Cases 6 and 7. The guide vanes show a small increase in percentage of total losses as Thoma number
drops, but decrease below 10% by Case 7.

Figure 3.40 shows the dimensional losses (in MW) through each stage, for both scales. It is clear now that for the high Thoma number cases (Cases 1 and 2), there are more draft tube losses in the prototype. However, for these cases there are still over 10% more total losses in the model, as the model has even higher losses in the guide vanes than the prototype has in the runner. As $\sigma$ is decreased the losses in guide vanes for both scales approach each other, and are roughly 0.2 MW in Case 7. The runner losses for both scales increase as cavitation breakdown occurs, and the losses in the prototype runner even become slightly larger than the model runner losses by Case 4. As with the guide vanes, the losses in both scales seem to converge as cavitation occurs to a greater extent. Cases 6 and 7 show increases in the draft tube losses for both scales, with a more abrupt increase in the model scale, which has much more vapor in this stage than the prototype. Therefore, the divergence in machine efficiency, seen between scales, is largely caused by the
divergence of losses in the draft tube, seen between scales, as cavitation breakdown occurs.

Recall that the model $\eta_{true}$ abruptly rises in Case 5, and becomes approximately equivalent with the prototype $\eta_{true}$ (Figure 3.32). Figure 3.40 exhibits that the model draft tube is mostly responsible for this effect. At this critical point the model draft tube performance improves. As even more cavitation forms (between Cases 5 and 7), the losses in the model draft tube increase sharply and the model runner losses remain relatively the same. A similar trend occurs in the prototype, although less abruptly and lagging slightly behind the model, due to the stronger influence of gravitational effects. It is not known whether the gradual improvement of the prototype’s draft tube performance between Cases 2 and 4 originates from similar physical mechanisms occurring in the model between Cases 4 and 5, or if enough cavitation has yet to form in the prototype draft tube for this effect to
occur. To answer this question, the prototype $\sigma$ would have to be decreased even further, or the physical mechanism causing this phenomenon would have to be uncovered.

### 3.5 Summary

The model and prototype steady-periodic results for single-phase and cavitating flow presented in this chapter provide two original contributions to the cavitating hydroturbine literature. For the first time, model-scale and prototype-scale cavitating hydroturbines were contrasted using multiphase CFD, to display steady-state performance differences associated with lack of $Fr$ similarity. In addition, a stage-by-stage performance decomposition of the cavitating hydroturbine, at both scales, was conducted for the first time. A formal mesh convergence study was presented
to validate the performance results.

This chapter shows that, as cavitation breakdown occurs, the experimental model tests do not depict the true prototype performance well at all, nor do they predict prototype vapor content accurately (especially at altitude locations farther away from the reference altitude). In fact, striking differences begin to arise in efficiency, and vapor far from the reference altitude, between the scales. This is a well-known phenomenon in the hydropower community. Nevertheless, the ability of multiphase CFD to successfully capture the breakdown and the differences between both scales, provides important information for running more successful model tests and possibly understanding the scaling of these flows. Furthermore, the results display that as cavitation breakdown occurs in this particular turbine, at a particular guide vane opening angle, that most of the losses occur in the draft tube. Draft tube losses continue to increase at the lowest Thoma numbers, while the runner and guide vane losses show signs of leveling off. More extensive cavitation breakdown simulations must be conducted at different design points to determine the decomposition of losses in a more complete manner. In conclusion, for examining steady-state performance at and between scales of cavitating hydroturbines, and analyzing the performance decomposition in such cases, multiphase CFD is a commercially applicable and beneficial supplement to model experiments.
4 Unsteady Methods and Approach

4.1 Overview

Steady-periodic computations are useful to study the steady-state performance behavior of cavitating hydroturbines, but they provide no means of analyzing the transient features of cavitation. Cavitation is an inherently unsteady and aperiodic phenomenon, and many aspects of the cavitating flow field can not be captured with steady-periodic simulations. Unsteady computations are expected to provide the dynamic behavior of the multiphase flow field, which is analyzed to allow for commentary about potential wear, erosion, and structural vibration due to collapsing vapor cavities. The details of the methods and approach to the unsteady full machine cavitating computations are provided in this chapter. First, the numerical approach is provided along with the turbulence and cavitation modeling methods. Then, the test case is described, which includes the entire machine rather than simply the guide vane, runner, and draft tube stages. Finally, details into the guidelines of unsteady data analysis in hydroturbines are discussed. For the unsteady computations, the Finite Volume Method (FVM) segregated flow solver of the STAR-CCM+ commercial software package is employed.

4.2 Segregated Approach to CFD

Just as density-based (i.e. coupled) CFD solvers were originally developed for compressible flow, pressure-based (i.e. segregated) CFD solvers were initially constructed for incompressible flow. In the development of the coupled approach, there was no direct concern with the pressure term in the momentum equations because, with compressible flow, that pressure is the thermodynamic pressure
(due to Stokes’ hypothesis (Stokes, 1845)). For isothermal compressible flow, the pressure can be determined from the density with a basic equation of state (i.e. \( p = p(\rho) \)). However, with incompressible flow, the mathematical expression of pressure which appears in the equations of motion is not related to the density at all, and becomes a function of just position and time (i.e. \( p = p(x, t) \)). In this regard, the mathematical construct of pressure becomes a Lagrange multiplier (Ferziger and Perić, 1996), or a mean reaction stress resulting from the constrained velocity field of incompressible flow (\( \nabla \cdot \mathbf{v} \)). Thus, in the incompressible flow formulation, the pressure has no separate explicit equation of its own, and must be solved for within the continuity and momentum equations, along with boundary conditions. Therefore, the pressure field becomes strongly coupled with the velocity field, and this presents difficulty in obtaining a numerical solution since there is not an independent equation for pressure (Ferziger and Perić, 1996).

4.2.1 Foundations of Projection Methods

The idea that the role of pressure in incompressible flow is simply to function as a mean reaction stress to the constrained velocity field, led to the development of segregated numerical solutions. By observing this idea from the opposite point of view, Chorin (1968) devised a projection procedure that decouples the pressure and velocity fields, and overcomes the primary numerical difficulties of incompressible flow. The projection method allows the pressure gradient to act as a force which constrains velocity, such that it satisfies the incompressible flow continuity equation. In Chorin’s projection method, beginning with a divergence-free velocity field, an intermediate velocity field \( \mathbf{v}^* \) is first computed from the momentum equations, ignoring the \( p \) term. There is no reason that \( \mathbf{v}^* \) must be divergence-free, since the term which ensures it will be so was ignored in its solving. Next, a pressure Poisson equation is derived by taking the divergence of the momentum equation at the next time step. The ‘source’ for the pressure Poisson is a function of \( \nabla \cdot \mathbf{v}^* \). In other words, the new \( p \) field is related to the part of \( \mathbf{v}^* \) which is not divergence-free (not incompressible). In effect, the ‘compressible’ part of \( \mathbf{v}^* \) is projected onto the space of \( \nabla p \) (Chorin and Marsden, 1990). The new \( p \), can be used in the momentum equation, to automatically constrain the velocity field, by way of \( \nabla p \), to satisfy continuity, and thus, solve for a corrected divergence-free velocity field.
The decoupling of the pressure and velocity fields, through the use of projection methods, is independent of the discretization method employed. Thus, segregated methods can just as easily be used with the Finite Element Method as they are with the Finite Volume Method. The projection method for fluid flows can be generalized into the following 3 simple steps:

1. **Prediction Step**: Predict an intermediate velocity field.

2. **Projection Step**: Solve a Poisson equation for a pressure related to the irrotational component of the intermediate velocity field.

3. **Correction Step**: Find the new solenoidal velocity field by constraining the intermediate velocity field, with $\nabla p$, to be divergence-free.

This general procedure, while expanded upon, and even successfully extended to compressible flow (Issa, 1986), remains at the heart of modern segregated flow solvers.

### 4.2.2 Governing Equations and Segregated Procedure

The general methodology of the STAR-CCM+ segregated flow solver for the locally homogeneous Eulerian multiphase mixture approach can be found in the STAR-CCM+ User Guide (CD-adapco, 2014), and in Mazaferija and Perić (1997, 1999); Mazaferija et al. (1998). A concise description of this approach is provided here. The conservation equations, in integral form, are

\[
\frac{\partial}{\partial t} \int_V dV + \int_S v_b \cdot n dS = 0 \tag{4.1}
\]

\[
\frac{\partial}{\partial t} \int_V \rho dV + \int_S \rho (v - v_b) \cdot n dS = 0 \tag{4.2}
\]

\[
\frac{\partial}{\partial t} \int_V \rho v dV + \int_S \rho v (v - v_b) \cdot n dS = \int_S (\tau - p I) n dS + \int_V \rho g dV \tag{4.3}
\]

\[
\frac{\partial}{\partial t} \int_V \rho \alpha_v dV + \int_S \rho \alpha_v (v - v_b) \cdot n dS = \int_V \dot{m} dV \tag{4.4}
\]

where $V$ is the control volume (CV) bounded by closed surface $S$, $n$ is the unit normal vector to CV surface $S$, $g$ is the gravitational acceleration vector, $I$ is the identity tensor, and $v_b$ is the velocity of a CV surface (when there is a moving...
The mass transfer per unit volume is $\dot{m}$ and the cavitation model defining its value is given in Section 4.2.5. Again, only two continuity equations are necessary, as only two distinct phases are present. In this formulation the mixture continuity (4.2) and the vapor phase continuity (4.4) equations are solved. Note that the liquid phase integral continuity equation must have a source term of $-\int_V \dot{m} dV$. Equation (4.3) is the vector mixture momentum equation, where the components of the viscous stress tensor $\tau$ are $\tau_{ij}$ given in (2.6). The Space Conservation Law (SCL) must also be satisfied for a moving mesh, and is shown in (4.1). All of the terms appearing in these equations, which were not mentioned here, are defined in Chapter 2.

The vapor phase continuity equation (4.4) is reformulated, such that it becomes a transport equation for just the vapor volume fraction. To do this, the terms are expanded and the following equation is obtained:

$$\frac{\partial}{\partial t} \int_V \alpha_v dV + \int_S \alpha_v (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS = \int_V \left( \frac{\dot{m}}{\rho_v} - \alpha_v \frac{D \rho_v}{Dt} \right) dV$$ (4.5)

where $\frac{D}{Dt}$ is the material derivative. For incompressible flow (i.e. constant phasic densities), as is studied throughout this dissertation, (4.5) becomes

$$\frac{\partial}{\partial t} \int_V \alpha_v dV + \int_S \alpha_v (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS = \int_V \frac{\dot{m}}{\rho_v} dV$$ (4.6)

A similar form can be derived for the liquid volume fraction. The mixture continuity equation (4.2) is reformulated as well, into its non-conservative form. This is done because the unsteady mixture density term ‘cannot be linearized in terms of pressure and velocity’, and acts as a large source term that causes numerical difficulties for the segregated approach (CD-adapco, 2014). To accomplish this, (4.6) and the corresponding transport equation for liquid volume fraction are summed together, while recalling that $\alpha_v + \alpha_l = 1$, to arrive at

$$\int_S (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS = \int_V \dot{m} \left( \frac{1}{\rho_v} - \frac{1}{\rho_l} \right) dV.$$ (4.7)

Note that, without the assumption of constant phasic densities, there would be a material derivative term still present on the right hand side of (4.7). Moreover, notice that constant phasic densities do not imply a divergence-free velocity field.
when mass transfer terms are present. Equation (4.6) is the form used to solve for the transport of the vapor volume fraction in the unsteady simulations, while (4.7) is used as the pressure-correction equation in these computations, and will be discussed further when the segregated procedure is covered.

Integration, Interpolation, and Differentiation

To perform the segregated FVM, the governing equations are discretized for each cell CV, to form a system of algebraic equations for each of (4.1)-(4.4). Then, each linear system is solved in a segregated fashion for each variable, treating all other variables as known (Muzafferija and Perić, 1999). Numerical approximations are required for spatial and temporal integration, interpolation from CV centers to CV face centers, and the calculation of derivatives. For the spatial integrals the simple 2nd-order midpoint rule is employed (Ferziger and Perić, 1996). With the volume integration, the midpoint rule approximates the integral as the product of the cell center value of the integrand and the discrete control volume. Likewise, with surface integration, the approximation gives the product of the cell face center value and the discrete cell face surface area. However, while the cell center values are known for volume integration, the cell face center values must be interpolated for surface integration. Additionally, the reconstructed field values at the cell faces, the integrand for the diffusive flux (due to the presence of the velocity gradient), the pressure gradient for pressure-velocity coupling, and the strain-rate and rotation-rate tensors for turbulence modeling, all require the calculation of gradients (CD-adapco, 2014).

Unlimited reconstruction gradients are computed from the cell center variable values (where an arbitrary scalar variable will be denoted by ϕ) at the cell centers. The unlimited reconstruction gradient of a variable at the center of cell 0 is denoted by \( (\nabla \varphi)_{r,0}^{ul} \). All of the gradients in the unsteady simulations are computed with the Hybrid Gauss-LSQ Method, which uses a scaling factor to blend the Gauss-Green and Least-Squares approaches, and is defined in CD-adapco (2014). \( (\nabla \varphi)_{r,0}^{ul} \) is unlimited because it does not bound the reconstructed variables on the cell faces by the minimum and maximum values of the cell centers associated with that cell face. A scaling factor \( \gamma_r \) is used to limit \( (\nabla \varphi)_{r,0}^{ul} \) so that the cell face values are bound by their neighboring cell center values. The particular limiter employed is that of Venkatatkrishnan (1993). The limited reconstruction gradient at cell 0 has
the form \((\nabla \varphi)^{lim}_{r,0} = \gamma_r \,(\nabla \varphi)^{ud}_{r,0}\). Limited reconstruction gradients are equivalent to the actual cell gradients (when using the Gauss-LSQ method) and can be used directly if the gradient is required. They are also used for interpolation to the cell faces to obtain the cell face values.

Since the convective fluxes are non-linear, the Picard-iteration procedure is utilized to linearize these terms, where the mass flux through the cell face \(\dot{M}\) is lagged (taken from the previous iteration). This results in the convective term being approximated at the cell faces as

\[
\int_S \rho \varphi \,(v - v_b) \cdot n \,dS = \bar{\varphi}_f \int_S \rho \,(v - v_b) \cdot n \,dS \approx \dot{M}_f \varphi_f
\]

(4.8)

where the over-bar represents an interpolated value and the subscript \(\cdot_f\) represents the cell face center point. \(\varphi_f\) is interpolated from the cell centers. Interpolation can be accomplished with a variety of schemes. Here, a hybrid 2\textsuperscript{nd}-Order Upwind/Bounded Central Difference (SOUBCD) scheme is employed. Considering two neighboring cells 0 and 1, sharing a cell face, the SOUBCD scheme is written as

\[
\dot{M}_f \varphi_f = \begin{cases} 
\dot{M}_f \left[ \vartheta \varphi_{f,0} + (1 - \vartheta) \,(\lambda \varphi_{f,0} + (1 - \lambda) \varphi_{f,1}) \right] & \text{for } 0 \leq \vartheta \leq 1, \dot{M}_f \geq 0 \\
\dot{M}_f \left[ \vartheta \varphi_{f,1} + (1 - \vartheta) \,(\lambda \varphi_{f,0} + (1 - \lambda) \varphi_{f,1}) \right] & \text{for } 0 \leq \vartheta \leq 1, \dot{M}_f < 0 \\
\dot{M}_f \varphi_0 & \text{for } \vartheta < 0, \vartheta > 1, \dot{M}_f \geq 0 \\
\dot{M}_f \varphi_1 & \text{for } \vartheta < 0, \vartheta > 1, \dot{M}_f < 0 
\end{cases}
\]

(4.9)

where \(\vartheta\) is a smooth, monotonic blending factor between the cell face values interpolated via 2\textsuperscript{nd}-order UD (the first term on the upper two RHS of (4.9)) and those interpolated via 2\textsuperscript{nd}-order CD (the second term on the upper two RHS of (4.9)). \(\vartheta\) is a function of the Normalized Variable Diagram (NVD) variable \(\vartheta\) (Leonard, 1988), where \(\vartheta\) is obtained from local flow values. Thus, \(\vartheta\) dictates the amount of blending between 2\textsuperscript{nd}-order UD and 2\textsuperscript{nd}-order CD, but allows the scheme to switch to 1\textsuperscript{st}-order upwind when convection boundedness criterion are not met (as in the lower two RHS of (4.9)). Therefore, hybrid SOUBCD can be thought of as providing a similar function to high-resolution flux-limiting described in Chapter 2, in that it allows for a higher-order scheme in smooth regions of flow but resorts to basic upwinding at local discontinuities to avoid unphysical oscillations. Again,
these unphysical oscillations can be an issue for bounded quantities such as $\alpha_v$, or positive-definite variables such as turbulent kinetic energy.

In Equation (4.9), $\lambda$ is a weighting factor based on the mesh stretching and the values of $\varphi_{f,0}$ and $\varphi_{f,1}$ are linearly interpolated from the cell centers using the limited reconstruction gradients, and are written as

\[
\begin{align*}
\varphi_{f,0} &= \varphi_0 + (x_f - x_0) \cdot (\nabla \varphi)_{r,0}^{lim} \\
\varphi_{f,1} &= \varphi_1 + (x_f - x_1) \cdot (\nabla \varphi)_{r,1}^{lim}
\end{align*}
\]

where $x$ is the position vector. SOUBCD is the default convection scheme for a Detached Eddy Simulation in STAR-CCM+ (which is the turbulence modeling approach to the unsteady results in this dissertation), and because of its CD component, it is beneficial for preserving the turbulent kinetic energy in cases where basic upwind schemes can cause unnaturally fast decay (CD-adapco, 2014). It should be noted that additional terms are also added to the interpolation schemes to correct for slight non-orthogonality in the mesh (Muzaferija and Perić, 1999).

Implicit temporal integration of the governing equations is performed with a 2nd-order scheme using the midpoint rule to approximate the integral (CD-adapco, 2014; Muzaferija and Perić, 1999). The scheme uses three separate time levels to compute a variable at the new time level, and integrates over an interval centered about the new time level (Muzaferija and Perić, 1999). Being implicit, this scheme is more stable than explicit, and larger time-steps can be used. The velocity of a CV surface $v_b$ does not need to be explicitly solved for, as the grid flux $G_f = v_b \cdot ndS$, due to grid motion, can be expressed by the volumes swept out by cell faces from one time-step to the next, in a manner that automatically satisfies the SCL (4.1) (Muzaferija and Perić, 1999). A 2nd-order scheme of the swept volumes between the last time-step, the current time-step, and the next time-step, is utilized to discretize $G_f$ (CD-adapco, 2014).

**SIMPLE Algorithm**

A commonly used, and highly successful implicit projection method for the solution of the N-S equations, known as the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm (Caretto et al., 1973), is utilized for the unsteady computations. SIMPLE consists of both outer and inner iterations which are looped
through in every time-step. The inner iterations are the iterations of the linear equation solver employed to solve each constant coefficient segregated equation. The outer iterations are the iterations of the SIMPLE projection procedure itself, that update the coefficient matrices and source vector with each cycle (Ferziger and Perić, 1996). Instead of using the actual pressure, as in the classical projection method, SIMPLE uses a pressure-correction instead. The pressure-correction \( p' \) is a small correction summed to the pressure from the last outer iteration. It is also used to determine the velocity-correction at the cell center \( \mathbf{v}' \), which is a small correction added to the intermediate velocity \( \mathbf{v}^* \), and similarly a cell face mass-flux-correction \( \dot{M}'_f \) to be added to the intermediate mass flux field \( \dot{M}^*_f \).

Before the SIMPLE algorithm is provided in more detail, the pressure-correction Poisson equation must be derived. Considering a single iteration of the algorithm (1 outer iteration) to be denoted by the superscript \( \cdot^m \), the cell center velocity field at the next SIMPLE cycle is calculated as

\[
\mathbf{v}^{m+1} = \mathbf{v}^* + \mathbf{v}'.
\]

(4.11)

By neglecting a velocity field of the momentum equation from which contributions of the pressure gradient have been removed (Ferziger and Perić, 1996), a relation between velocity and pressure correction terms is given as

\[
\mathbf{v}' = -\frac{V \nabla p'}{a^v}.
\]

(4.12)

Taking the divergence of (4.11), and utilizing the mixture continuity equation (4.7) and the relation between velocity and pressure correction terms (4.12), the pressure-correction Poisson equation is obtained:

\[
\nabla \cdot \left( \frac{V \nabla p'}{a^v} \right) = \nabla \cdot \mathbf{v}^* - \dot{m} \left( \frac{1}{\rho_v} - \frac{1}{\rho_l} \right).
\]

(4.13)

Here, \( V \) is the cell volume and \( a^v \) is the vector of central coefficients for the discretized linear velocity equation system. Note, that the gradient of the pressure-correction must use the discretization method from the momentum equation, while the divergence must be discretized is the same manner as the mixture continuity equation (Ferziger and Perić, 1996). Equation (4.13) is used to solve for the pressure-correction in the SIMPLE projection method.
The steps of SIMPLE are now outlined. First, the time is advanced to the
next time-step. Then, the mesh is moved to its new position in the case where it
is required to be in motion. Next, one outer iteration of the SIMPLE algorithm
(denoted by superscript \(m\)), where all segregated linear equation systems are each
solved for a single unknown by assuming all other variables in the equation system
are known (taken from the old values), proceeds as follows:

1. Boundary conditions are set.
2. Reconstruction gradients are computed and limited. Cell gradients are ob-
tained.
3. Cell face center values are interpolated for flux terms.
4. Momentum equation (4.3) solved for in discretized/linearized form to obtain
intermediate velocity field \(v^*\).
5. Intermediate mass fluxes at cell faces \(\dot{M}_f^*\) are computed.
6. Discretized pressure-correction Poisson equation (4.13) is solved to obtain
   cell center values of \(p'\)
7. Pressure field corrected: \(p^{m+1} = p^m + \gamma_p p'\) where \(\gamma_p\) is the pressure under-
   relaxation factor. Update the boundary pressure corrections.
8. Cell face mass fluxes corrected: \(\dot{M}^{m+1}_f = \dot{M}_f^* + \dot{M}'_f\)
9. Cell center velocities are corrected using (4.11) and (4.12).
10. Volume fraction transport equation (4.6) solved in discretized form to obtain
    \(\alpha^{m+1}_v\). Turbulence transport equations solved as well.
11. Other values updated (liquid volume fraction, mixture density, cavitation
    model mass transfer, etc.)

Outer iterations are repeated until the residuals are reduced by the desired tolerance.
Then the time is advanced by time-step \(\Delta t\), and the entire algorithm is repeated
again. Note that, using a colocated variable arrangement, checker-boarding can
occur in the pressure field (Ferziger and Perić, 1996). To overcome this, the Rhie-
Chow correction term (Rhie and Chow, 1983) is subtracted from the interpolated
normal cell face intermediate velocities, to detect check-boarding and smooth it out (Muzaferija and Perić, 1999).

Explicit under-relaxation of the pressure-correction only allows the old pressure to be updated by a factor $\gamma_p$ of the pressure-correction, reducing instabilities. Variables such as $v$, $\alpha_v$, and turbulence variables are under-relaxed as well. Although, unlike the explicit under-relaxation of the pressure-correction, the under-relaxation factors for these variables are implicitly included in the linear equation system that is utilized to solve for them (CD-adapco, 2014). Implicit under-relaxation of this type, first proposed by Patankar (1980), increases diagonal dominance of the coefficient matrices, and is more efficient than explicit under-relaxation (Ferziger and Perić, 1996).

### 4.2.3 Volume of Fluid Method

A locally homogeneous mixture approach, along with SOUBCD and limited reconstruction gradients, allow for a similar approach for capturing the interface between fluids as is shown in Section 2.2.4. Thus, this process is only a pseudo-VOF method in that it reverts to 1st-order UD across the discontinuities induced by the spatial distribution of vapor and liquid. STAR-CCM+ has the ability to enhance this approach, resolving a more accurate interface, by using an interface sharpening term added onto the volume fraction transport equation (CD-adapco, 2014), and a High-Resolution Interface Capturing (HRIC) convection discretization scheme (Muzaferija and Perić, 1999). However, the sharpening factor is set to zero, and the HRIC scheme is not used in this dissertation, and thus, 1st-order UD is used for convection of volume fraction across interfaces. This approach is deemed adequate for the current level of investigation into cavitating hydroturbines, as the majority of the cavities seen in the experiments are exceptionally ‘cloudy’, and thus, do not display well-defined large-scale sharp interfaces. In addition, natural cavitation is primarily driven by the mass transfer rate $\dot{m}$, and the order of the volume fraction convection scheme across the interface is not as important to capturing the dynamics.
4.2.4 Detached Eddy Simulation

A Delayed Detached Eddy Simulation (DDES) turbulence modeling approach is utilized for the unsteady computations, primarily to permit the capturing of large-scale turbulent structures in important regions, while maintaining the computational efficiency of unsteady RANS in boundary layers and other less interesting regions of flow. A large reduction in computational effort is obtained in the RANS regions over the effort required for a full LES in the entire domain. In zones where the mesh is adequately resolved to handle LES, DES allows for it. While RANS causes the eddy viscosity to unphysically dominate the dissipation (even with mesh refinement), LES reduces the effects of eddy viscosity by resolving the large-scale structures and only modeling the sub-grid scale Reynolds stresses. Even though modeling is still required for the sub-grid scale Reynolds stresses, LES captures the scales whose ‘strength and size make them by far the most effective transporters of the conserved properties’ (Ferziger and Perić, 1996). DES has proven to be beneficial in accurately capturing the transport of vapor in cavitating flow (Kinzel, 2008, Kinzel et al., 2007). Thus, it may prove to be advantageous in capturing the unsteady dynamics of the industrial-scale cavitating flows that occur in hydroturbines, at a greatly reduced computational cost from LES.

DES was originally formulated with the single-equation RANS Spalart-Allmaras (S-A) turbulence model (Spalart et al., 1997), and a form of the S-A model is used here as well. The S-A model solves a transport equation, shown in (4.14) in integral form, for the ‘modified diffusivity’ \( \tilde{\nu} \).

\[
\frac{\partial}{\partial t} \int_V \rho \tilde{\nu} dV + \int_S \rho \tilde{\nu} (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS = \\
\frac{1}{\tilde{\varsigma}} \int_S (\mu + \rho \tilde{\nu}) \nabla \tilde{\nu} \cdot \mathbf{n} dS - \frac{C_{\mu 2}}{\tilde{\varsigma}} \int_V (\mu + \rho \tilde{\nu}) \nabla \tilde{\nu}^2 dV + \int_V G_\varphi - \Upsilon_\varphi dV \quad (4.14)
\]

In Equation (4.14), the terms \( G_\varphi \) and \( \Upsilon_\varphi \) represent the production and dissipation of turbulence, while the first two terms on the RHS are the combined conservative and non-conservative diffusions. Appendix A provides the full details of these terms and the complete S-A DDES turbulence model to allow for brevity in this section. For a RANS calculation, Equation (4.14) is solved for \( \tilde{\nu} \) and the eddy viscosity can be computed with

\[
\mu_T = \rho \tilde{\nu} f_\nu \quad (4.15)
\]
where \( f_{\nu 1} \) is a function of \( \tilde{\nu} \) and the kinematic viscosity \( \nu \) and is provided in Appendix A. Thus, the S-A model provides the eddy viscosity for a RANS computation and can also be used with wall functions.

To compute the large eddies only, as is done in LES, the filtered Navier-Stokes equations must be used. Here, the velocity is filtered by a localized kernel function that has a length scale \( \Delta \) associated with it (Ferziger and Perić, 1996). If eddies are larger than \( \Delta \), then they will be resolved by the filtered N-S equations. \( \Delta \) is the largest distance between the computational cell center under consideration and the cell center of any of its neighboring cells. If eddies are smaller than \( \Delta \) then they must be modeled by a sub-grid scale (SGS) model. The sub-grid scale terms can be represented as a hypothetical stress, in a similar manner to the RANS turbulence models. The SGS model used in this work is that of Smagorinsky (1963). The SGS eddy viscosity is represented as

\[
\mu_T = \rho \Delta^2 S, \tag{4.16}
\]

where \( S = \sqrt{2S: \bar{S}} \) and \( S = \frac{1}{2} \left( \nabla v + \nabla v^T \right) \). DES links RANS and LES together by solving the momentum equation (4.3) using the eddy viscosity determined from (4.15) (obtained by solving (4.14)) in RANS regions, and using the eddy viscosity determined from (4.16) in LES regions. This feat is accomplished through the turbulent length scale \( \tilde{d} \) (given below), that modifies the wall distance \( d \) and determines whether the mesh (and flow in the case of DDES) dictate a region as RANS or LES.

\[
\tilde{d} = d - f_{\text{dmax}} (0, d - \Psi C_{DES} \Delta) \tag{4.17}
\]

\( C_{DES} \) is a parameter capable of tuning DES and is commonly set to 0.65, where simulations have proven to be insensitive to this parameter (Strelets, 2001). \( \tilde{d} \) dictates whether the eddy viscosity will be based on the Reynolds stresses (4.15) or the SGS Reynolds stresses (4.16). In this dissertation, the DDES formulation is used to improve the ability of conventional DES to differentiate between RANS and LES regions (Spalart et al., 2006). DDES uses the function \( f_d \) to accomplish this, and it is defined in Appendix A. DDES allows \( \tilde{d} \) to depend on eddy viscosity and the velocity gradients, and thus, the flow itself, when determining whether a region of the mesh should use RANS or LES. Finally, the term \( \Psi \) is a low-\( Re \) correction function that prevents activation of low-\( Re \) terms when in LES mode.
4.2.5 Cavitation Modeling

A seed-based mass transfer model is utilized for cavitation modeling in the unsteady computations. Seed-based models assume that the vapor phase is present in each control volume as spherical bubbles. They develop relations for mass transfer rates (per unit volume) which are a function of bubble radius $R$ and the rate of change of bubble radius as the bubble flows. Here, the mass transfer rate for the seed-based model is defined as

$$\dot{m} = n_0 \alpha l 4\pi \rho \dot{R}$$

(4.18)

where $n_0$ is the seed density (number of bubbles/unit volume). Both $n_0$ and a minimum value of $R$ are specified as inputs into the model. The radius can be defined as a function of vapor volume fraction by

$$R = \left[ \frac{\alpha v}{n_0 \frac{4}{3} \pi (1 - \alpha v)} \right]^{1/3}.$$

(4.19)

The material derivative of $R$ must be determined by a model. The model is based on the Rayleigh-Plesset (R-P) equation which is a second-order ordinary differential equation that governs the behavior of the radius of a single spherical bubble through time. It is derived from the incompressible, isothermal, spherically symmetric, radial momentum N-S equation (Brennen, 2013). The R-P equation, with viscous and surface tension effects is

$$R \frac{d^2 R}{dt^2} + 3 \left( \frac{dR}{dt} \right)^2 = \frac{p_v - p_\infty}{\rho_l} - \frac{4\mu_l}{\rho_l R} \frac{dR}{dt} - \frac{2s_\sigma}{\rho_l R}$$

(4.20)

where the two final terms on the right hand side are due to viscous effects from the dynamic boundary condition at the bubble surface (not from the momentum equation itself) and the surface tension effects from surface tension $s_\sigma$. In seed-based models these terms are commonly ignored. $p_\infty$ is the local pressure of the liquid surrounding the bubble. The Schnerr and Sauer (2001) inertia-controlled bubble growth model utilized in this work, simplifies the R-P equation and defines the
bubble growth rate as

\[
\frac{DR}{Dt} = \left[ \frac{2}{3} \frac{p_v - p_\infty}{\rho_l} \right]^{1/2}.
\]  

(4.21)

The bubble growth rate \( \frac{DR}{Dt} \) can be scaled by a factor such that positive and negative bubble growth rates can vary. However, in this dissertation the positive and negative scaling factors were both chosen to be unity.

### 4.2.6 Unsteady Solution Strategy

Unsteady cavitating flow in the fully-coupled model and prototype scale hydroturbines is computed using STAR-CMM+’s finite-volume segregated VOF solver for a locally homogeneous, constant phasic density, Eulerian mixture. Turbulence is modeled with a Delayed Detached Eddy Simulation based on the Spalart-Allmaras one-equation turbulence model with 2\(\text{nd}\)-order convection. Cavitation modeling is based on the seed-based R-P approach to mass transfer modeling. Mesh motion is accounted for in the governing equations. Limited reconstruction gradients are computed from the cell center values to obtain the cell gradients, as well as to interpolate to the cell faces in preparation for integration of the flux terms. Interpolation of the convective flux is conducted with the 2\(\text{nd}\)-order Upwind/Bounded Central Differencing scheme which adequately preserves the turbulent kinetic energy in smooth regions of flow, but reverts to 1\(\text{st}\)-order UD across sharp gradients to maintain boundedness of variables. Spatial integration is accomplished with the midpoint rule, and the convective fluxes are linearized by Picard-iteration (utilizing the mass fluxes from the previous iteration). For each time-step, the SIMPLE projection method dictates the outer iterations by first predicting an intermediate velocity, finding a pressure-correction, correcting pressure and velocity, and then solving the volume fraction and turbulence transport equations. The inner iterations of each segregated equation use an Algebraic Multi-Grid (AMG) linear solver (Ferziger and Perić, 1996; Stüben, 2001), which accelerates Gauss-Seidel iteration by conducting correction sweeps on coarse meshes (CD-adapco, 2014). Time integration employs a 2\(\text{nd}\)-order method, and after the outer iterations are complete, the solver proceeds to the next time-step.
4.3 Fully-Coupled Francis Hydroturbine Case

The test case remains as the same Francis turbine that was described in Section 2.3 for the steady-periodic results. However, to capture the true unsteady flow in the turbine, the periodic assumption is no longer valid. Thus, the single-blade passages for the guide vanes and the runner must be rotated to complete the full wheel. Furthermore, besides the draft tube (which was present as an aperiodic stage in the steady results), the other aperiodic components are included as well. Therefore, the entire Francis hydroturbine machine that was tested with the experiments is contained within the computational domain of the unsteady simulations.

4.3.1 Geometry and New Mesh Components

The entire machine geometry consists of a penstock, partial spiral casing, stay region, guide region, runner region, and draft tube. Figure 4.1 shows that the penstock has four inlets of equivalent area. These inlets are where the flow enters from the headrace of the dam. The flow from three of the inlets converges downstream of structural supports and enters into the partial spiral casing. The flow from the other inlet is completely disjoint from the partial spiral casing, but it does turn the flow slightly before it enters the stay region. The stay region is aperiodic due to the disjoint penstock. The blades are of the same periodicity as the guide vanes, but there are 18 rather than 20, because of walls from the penstock which continue into the stay region. These walls are one of the reasons why the stay vanes are not included in the steady-periodic computations. Next, the flow encounters 20 stay vanes followed by 11 runner blades. Finally, the fluid travels through the draft tube and exits where it would emerge into the tailrace of the dam. The static pressure contours of Figure 4.1 show that hardly any pressure is lost between the penstock inlets and the guide region entrance, and thus there is not much specific energy lost. This helps validate the assumption, which was made in the steady-periodic computations; that the net specific energy of the entire machine can be used to determine the total pressure at the guide region entrance without much loss in prediction of machine performance.

The same coarse, medium, and fine fully-structured meshes of the steady-periodic computations for the guide vane, runner, and draft tube are utilized for
the unsteady computations as well. To accomplish this, the guide vane mesh was rotated 20 times about the axis of rotation, while the runner mesh was rotated 11 times. Meshes had to be constructed for the penstock, partial spiral casing, and stay regions. STAR-CCM+'s auto-generation polyhedral mesher was employed to construct the volume mesh of the penstock/partial spiral casing, alongside the prism layer mesher to build quality wall-function resolution (CD-adapco, 2014). Pointwise (Pointwise, 2011) was used to build a fully-structured mesh for the stay region. A coordinate-slice displaying these meshes is shown in Figure 4.2. A close-up of the prism layer in the penstock and a closer view of the stay vane mesh are shown in Figure 4.3. A mesh for a plenum (not pictured) was included after the draft tube exit. This was done to help maintain stability of the simulations.

Meshing guidelines for DES were set forth by Spalart (2001) where RANS and LES grid zones were discussed in detail. The current mesh (guide vane through draft tube) is already proven to capture RANS flow. However, alternative considerations are required for the LES regions. In the focus regions (i.e. near-wake zones where turbulence must be well resolved) Spalart (2001) ideally advocates for isotropic spacing, in terms of better performance and to only filter out statistically isotropic
turbulence. The focus region does not extend far downstream, and the mesh can transition to non-isotropic cells outside of the near-wake even though this region is still an LES zone. The fine mesh, which is used for all of the mesh/time-step converged results, is fairly isotropic in focus regions where major cavitation is forming (leading-edge suction-side of the runner blade and just beneath the hub nose cone), and mesh/time-step convergence is accomplished for the variables under investigation.

Table 4.1 provides the number of cells contained within each component of the
Table 4.1: Number of cells within each stage for the coarse, medium, and fine meshes. Note that each total includes a $0.4 \times 10^6$ cell plenum as well.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>$2.6 \times 10^6$</td>
<td>$33.3 \times 10^6$</td>
<td>$27.6 \times 10^6$</td>
<td>$24.2 \times 10^6$</td>
<td>$1.4 \times 10^6$</td>
<td>$89.5 \times 10^6$</td>
</tr>
<tr>
<td>Med.</td>
<td>$2.6 \times 10^6$</td>
<td>$33.3 \times 10^6$</td>
<td>$46.2 \times 10^6$</td>
<td>$41.1 \times 10^6$</td>
<td>$2.0 \times 10^6$</td>
<td>$125.6 \times 10^6$</td>
</tr>
<tr>
<td>Fine</td>
<td>$2.6 \times 10^6$</td>
<td>$33.3 \times 10^6$</td>
<td>$77.4 \times 10^6$</td>
<td>$71.3 \times 10^6$</td>
<td>$3.1 \times 10^6$</td>
<td>$188.1 \times 10^6$</td>
</tr>
</tbody>
</table>

machine for the coarse, medium, and fine meshes, alongside the total number of cells for each level of mesh refinement. It is clear that only the guide vane, runner, and draft tube regions are refined. The other regions maintain the same amount of cells for each level of refinement. Thus, the representative cell length of each level of refinement is equivalent to the steady-periodic cases, if only the guide vane, runner, and draft tube are considered for its calculation. The intricacies of the flow in the penstock and stay vanes are assumed to have little significance on the flow downstream of the guide vanes, and it is assumed that their lack of refinement is not critical to the mesh convergence study.

### 4.3.2 Boundary Conditions

The BC remain similar to Section 2.3.3 other than the fact that periodic boundaries and mixing planes are unnecessary here, and are not used. The surfaces of revolution, which defined the mixing planes (at the inlet and outlet of the runner) in Chapters 2 and 3, are now boundaries between the rotating runner mesh and the stationary guide vane and draft tube meshes. A sliding-mesh interface represents the boundary condition between the stationary and rotating components. The runner mesh rotates at rate $\Omega$ with respect to the inertial reference frame of the stationary components. As the runner mesh rotates, the sliding-mesh interface must update at each time step. This procedure is a large percentage of the computational time for the unsteady simulations, usually taking fairly more time than the outer and inner iterations of the flow solution at a given time step. The updating of the sliding-mesh interface consumes more computational time as the mesh is refined.

The inlet, now at the true machine inlet, is still defined with the total pressure, based on the net specific energy from the experiments, and a flow angle. Since the penstock inlet surfaces have uniformly oriented normal vectors (in a cartesian
coordinate system), the flow angle is simply specified as normal to the boundary. The turbulent boundary layers are again modeled with blended (all $y^+$) wall functions as in Section 2.3.3 and display similar values of $y^+$ to those reported in Table 2.2 for both the model and prototype.

### 4.3.3 Initial Conditions and Convergence

To initialize the full-machine simulations, the steady solution is first computed with a ‘frozen rotor.’ In other words, a steady computation is conducted with the runner at a fixed location and without the use of circumferential averaging at the inlet and exit of the runner. This is not physically accurate, but it does provide a quality flow field for the initialization of unsteady computations. Next, the unsteady single-phase simulations are conducted and run until the start-up transient has subsided. Finally, the cavitation modeling is switched on, the multiphase simulation is run until the start-up transient has subsided (it usually takes a couple revolutions), and then run a few more revolutions to allow for accurate unsteady data. The residuals converge by approximately one order of magnitude (and sometimes more) at each time step, which is considered acceptable for unsteady simulations using the SIMPLE method (Muzaferija and Perić, 1999). Each time-step consists of 20 outer iterations in order to reduce the residuals by a respectable amount.

Besides a mesh refinement study, a time-step refinement study is also conducted. The fine mesh solution is computed with three different $\Delta t$ values (See Table 4.2). Each $\Delta t$ is halved from the $\Delta t$ larger than it. The largest $\Delta t$ has 20 time-steps between each runner blade-guide vane passage. In comparison, the smallest $\Delta t$ has 80 time-steps per blade-vane passage, while the medium level $\Delta t$ has 40 time-steps per blade-vane passage.

#### Table 4.2: Three $\Delta t$ values and number of time-steps per revolution.

<table>
<thead>
<tr>
<th>$\Delta t$ [s]</th>
<th>Time-steps rev [#]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0024</td>
<td>400</td>
</tr>
<tr>
<td>0.0012</td>
<td>800</td>
</tr>
<tr>
<td>0.0006</td>
<td>1600</td>
</tr>
</tbody>
</table>
4.3.4 Computational Resources

Each unsteady cavitating CFD solution is computed on the Department of Defense High Performance Modernization Program (DoD HPCMP) Cray XC-30 (1.2 PFLOPS) named Lightning, belonging to the U.S. Air Force Research Laboratory (AFRL). Lightning contains 2370 compute nodes with 64 GB of memory each. Each node has 24 Intel Xeon E5-2697v2 cores, each with a speed of 2.7 GHz. Each case is computed on 1680 cores (70 nodes), and takes approximately 144 hours (6 days) for 3 runner revolutions at the smallest time-step and with the finest mesh.

4.4 Unsteady Model Experiments

Unfortunately, there is no unsteady experimental data available for the test case studied in this dissertation. However, there are particular suggestions for unsteady measurements provided by IEC (1999) and Dörfler et al. (2013). If unsteady measurements are conducted, the primary variables are commonly the fluctuating pressure and shaft torque. It is recommended that the minimal locations of pressure probes be on two sides of the draft tube near the exit of the runner (preferably on the elbow-side and opposite to the elbow-side), and in the penstock. Analysis is most commonly presented in the time and frequency domains. The time signals are analyzed with visualization of the periodic or stochastic nature of the fluctuations, and relative peak-to-peak amplitude calculations. The dominant frequencies are then investigated in the amplitude spectra which are obtained by performing a fast-Fourier transform (FFT) on the time series using a Hanning window with overlap. The amplitude spectra are usually displayed versus the normalized frequency to express the number of fluctuations per revolution. Here, the normalized frequency is expressed as $\omega/\Omega$, where $\omega = 2\pi f$ ($f$ is frequency), and $\Omega$ is the rotation rate of the runner. Normalized frequency has units of fluctuations per revolution.

As stationary data is only obtained for roughly 3 runner revolutions, there is not much of a sample size for spectral analysis of the CFD results, when compared with typical spectral analysis of experimental results. Thus, the sample size is too small to accurately analyze some low-frequency components of the machine in the frequency domain. For instance, 3 revolutions is typically 1 fluctuation induced by the precessing vortex core in the draft tube, and there is no way to determine if
the simulation of this feature has reached a periodic state, nor is the sample size long enough to analyze its fluctuating behavior. However, the spectral analysis can still prove somewhat accurate and beneficial to the analysis of higher frequency components which have more than just a few fluctuations over 3 revolutions (i.e. greater than 5/rev). Furthermore, the frequencies of the fluctuations which are fairly forced by the constant rotation rate of the runner (i.e. runner blade-guide vane interaction) should be very accurate. The FFT of each time-series is taken to allow for the minimum frequency resolution $\Delta f$, so that the spectra usually have a $\Delta f$ between 0.3 fluctuations/revolution and 0.5 fluctuations/revolution. Although the FFT of the results have a maximum frequency of over 800/rev, frequency content above 60/rev is always at lower amplitude than the dominant sub-60/rev fluctuations, and is usually just harmonics of the sub-60/rev oscillations. Thus, spectra are predominantly presented for frequencies up to 60/rev.

When the model system has no significant external interactions that do not occur in the prototype, then the pressures and torques are transferable from model to prototype. Due to the nature of model test loops, for which the model does interact with external systems, and the fact that the state-of-the-art does not allow for accurate quantification of these interactions, the unsteady model measurements are primarily for qualitative information only (IEC, 1999). Additionally, if $Fr$ similarity is not fulfilled in cavitation tests, as it most commonly is not, then the pressure fluctuations would not be transferable to the prototype. While the unsteady model CFD results presented here do not have the ability to model external sources that occur in the model experiments, both the model and prototype computations have the benefit of not having any interaction with external systems. This allows for an investigation of the differences between model and prototype unsteady fluctuations based on factors such as the lack of $Re$ and $Fr$ similarity.

The time-averaged unsteady performance results are compared with the steady-state performance results in an attempt to display the accuracy of the unsteady simulations. As there is no unsteady experimental data to compare with, a mesh and time-step refinement study (using time-averaged and unsteady variables of interest) is conducted to show convergence, in an effort to validate the results.
5 Unsteady Results and Discussion

5.1 Introduction

Unsteady cavitating DES of the entire model and prototype hydroturbines are conducted to display the efficacy of CFD for capturing time-dependent multiphase phenomena in not just the draft tube, but in the runner/guide vane regions as well. Model and prototype scales are studied to show the differences in torque and pressure fluctuations caused by lack of $Re$ and $Fr$ similitude. Furthermore, maximum pressures on the runner blade surfaces are analyzed to provide quantitative data on high-pressure cavitation collapse phenomena, and allow for qualitative commentary on potential erosion to the blades. Visualizations of cavitation collapse on the runner blades are presented along with an image series depicting the shedding and collapse of the cavitating hub vortex.

To collect data from the simulations, surface forces are monitored on critical turbine components, a number of pressure probes are placed within the turbine, and other key physical and numerical simulation variables are tracked through time. To obtain the torque of each individual runner blade, as well as the net torque on the runner itself, the pressure and viscous forces are obtained at each cell face on the surfaces for each time-step of the simulation. The total pressures and flow rates are monitored at the inlet surfaces of the penstock and the outlet surface of the draft tube. With this information, and the net torque on the runner, the machine efficiency is obtained at each time-step. The torque on each individual runner blade is presented as a percentage of the mean (time-averaged) torque on all 11 blades, while the net torque on the runner is shown as a percentage of the mean net torque on the runner. Furthermore, the mean torque is subtracted from the unsteady torque before normalization to present the normalized torque fluctuations.
as percentages of the mean, and to reveal the low-frequency components which would otherwise be overshadowed by the mean-value at 0/rev. The normalized torque $T_E$ is given by

$$T_E = \frac{T - \bar{T}}{\tau_{net}}$$

where time-averaged quantities are represented with an over-bar.

Pressure data is collected with six probes in the draft tube and two probes in either disjoint penstock region. The locations of these probes are shown in Figures 5.1 and 5.2. Note that the ‘probes’ in the penstock regions are actually surfaces within the flow field that the pressure is spatially averaged over, while the probes in the draft tube are at physical points on the draft tube wall. The results presented in this dissertation are pressures taken from probe-1 (first probe on the left after the entrance to the draft tube), probe-2 (first probe on the right after the entrance to the draft tube), probe-A (surface in the penstock with the partial spiral casing), and probe-B (surface in the penstock without the partial spiral casing). The pressure is normalized by the net specific energy across the machine, with a method similar to the torque. Thus, the presented pressures are in units of the percentage of fluctuation relative to the net specific energy. The normalized pressure $p_E$ is given by

$$p_E = \frac{p - \bar{p}}{\rho_1 E}.$$  \hfill{(5.2)}

The chapter is laid out in the following manner. Initially, a mesh and time-step
convergence analysis of a representative case is provided, in an attempt to show the
degree to which all of the following presented cases have converged. Time-averaged
performance data from the unsteady computations is then studied and compared
with experimental model data, standard scalings from the experiments, and the
steady-periodic results. Next, some interesting features, such as a shallow draft tube
induced runner load imbalance, and the captured precessing vortex core (PVC), are
discussed. Fluctuating torque time-series’ and their amplitude spectra are displayed,
and the differences between single-phase, cavitating, model, and prototype flows
are highlighted. The same procedure is then repeated with pressure fluctuations in
the draft tube and penstock. Finally, the maximum blade pressures are analyzed
and potential erosion is considered.

5.2 Convergence

A convergence study, by means of mesh and time-step refinement, is conducted, and
the convergence of time-averaged and unsteady variables of interest are presented.
The time-averaged variables of interest are the machine efficiency and the volume of
vapor in the entire computational domain. These are similar variables of interest to
those from the steady-periodic mesh convergence study in Section 3.3.1. Extending
the convergence study to unsteady variables, further variables of interest are
analyzed. The peak-to-peak amplitudes and amplitude spectra, for both mesh and
time-step refinement, are investigated. When appropriate, ‘exact’ values are shown
via Richardson extrapolation. As running all of the cases with every mesh and
time-step is very computationally expensive, model-scale Case 2 ($\sigma = 0.99$) is chosen as a representative case for the convergence analysis. For the mesh refinement, this case is simulated at the smallest time-step only, while for the time-step refinement, this case is computed with the finest mesh only.

5.2.1 Time-averaged Variable Convergence

Stable results of time-dependent computations, using explicit procedures, require a convective Courant number $Co_i = \frac{v_i \Delta t}{\Delta x_i}$ of less than one. This can severely restrict the time-step of the solution. However, the SIMPLE procedure employed for the presented unsteady computations, is semi-implicit, and allows $Co \gg 1$ while maintaining stability. Thus, the time-step can be set to larger values and accurate results can be obtained for regions where $Co$ is low enough to resolve the unsteady flow (i.e. most regions in the computational domain). Accurate results for regions where $Co$ is large require a smaller time-step.

In Table 5.1 the values of the maximum convective Courant number $Co_{max}$ and mesh Courant number $Co_{mesh}$ are given for each mesh and time-step. $Co_{max}$ increases with mesh refinement at constant $\Delta t$, as expected, since $\Delta x$ decreases. Likewise, at constant $\Delta x$, $Co_{max}$ decreases with time-step refinement. With the finest mesh, and smallest $\Delta t$, the convective Courant number is roughly 100, and thus, much greater than 1. While the maximum convective Courant number represents the region where the solution is least accurate, it should be noted that the mean convective Courant number, of each stage and the entire domain, is well below 1 in every case. The large values of $Co$ are only located in a small region at the sliding-mesh interface of the guide-vane and runner, near the shroud. Moreover, in all regions away from the walls, where a LES is being employed, the maximum Courant number is below one in all cases.

To calculate the Courant number of the rotating mesh, the average circumferential grid spacing at the runner inlet interface on the shroud side, is used for $\Delta x$. The circumferential velocity of the mesh at this radius is given as $\Omega R$. $Co_{mesh}$ has values below 4 for all refinement cases, and for the finest mesh and smallest time-step, it is below 1.

Figure 5.3 shows mesh and temporal convergence of the time-averaged machine efficiency $\eta$. Mesh refinement displays asymptotic convergence for which a Richard-
Table 5.1: Maximum convective and mesh Courant numbers for each mesh and time-step.

<table>
<thead>
<tr>
<th>Case</th>
<th>σ</th>
<th>Mesh</th>
<th>Δt</th>
<th>$C_{o_{max}}$</th>
<th>$C_{o_{mesh}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.99</td>
<td>Coarse</td>
<td>0.6</td>
<td>74</td>
<td>0.65</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Medium</td>
<td>0.6</td>
<td>101.5</td>
<td>0.80</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>0.6</td>
<td>102</td>
<td>0.97</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>1.2</td>
<td>185</td>
<td>1.94</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Fine</td>
<td>2.4</td>
<td>355</td>
<td>3.87</td>
</tr>
</tbody>
</table>

Richardson extrapolation can be conducted. Values between the coarse and fine mesh do not even change by half a percent, and thus excellent mesh convergence is displayed. With decreasing time-step, $\eta$ shows oscillatory convergence and does not lend itself to Richardson extrapolation. Oscillations with reduced time/mesh refinement do not imply that the variable of interest is not converged, as the variable could be very close to convergence and out of the asymptotic range (Roache, 1998). Here, because $\eta$ is hardly changing with time-step refinement, it is believed that the variable is converged. Note that the converged $\eta$, of roughly 0.79, is in error of 6% from the experimental results, which is fairly accurate but not as accurate as the steady-periodic results (4% error). This will be discussed further when the time-averaged results are presented.

Figure 5.4 displays the mesh and temporal convergence of the time-averaged vapor volume fraction in the entire computational domain. Asymptotic convergence is seen for both the mesh and time-step refinement, and thus, Richardson extrapolation is possible, and shown. Both the mesh and time-step refinement converge to a similar extrapolated value of volume of vapor $V_v$. It is also of note that the time-averaged minimum pressure in the entire computational domain is always only a few kPa lower than the vapor pressure $p_v$ in every computation presented in this chapter. Additionally, the pressure of the vapor itself hardly deviates from $p_v$ in all of the simulations.
Figure 5.3: Convergence of time-averaged machine efficiency.

(a) Mesh Convergence

(b) Temporal Convergence

Figure 5.4: Convergence of time-averaged $V_v$ in computational domain.
5.2.2 Unsteady Variable Convergence

An attempt is now made to display the convergence of key fluctuating variables which are used to analyze the complete unsteady results. First, the mesh and time-step refinement study of the torque on blade-1, are shown in Figure 5.5. The peak-to-peak amplitudes of the normalized torque fluctuations display oscillatory and asymptotic convergence with mesh and time-step refinement, respectively. Note that these peak-to-peak values do not change much at all with refinement, and remain between 1.6 and 1.8 percent of the mean torque on all 11 blades, for all instances. The amplitude spectra allow the visualization of the frequency content for each level of refinement. It is clear that the dominant frequencies, and their amplitudes, hardly change with refinement.

Figure 5.6 presents the mesh and time-step refinement study of the normalized pressure fluctuations at probe-1 in the draft tube. The peak-to-peak amplitude does not change much between the coarse and medium meshes, but for the fine mesh, it displays a sharp increase. This occurs because a new dominant frequency (7.33/rev) is present in the fine mesh results that is thought to be caused by the amount of vapor in the draft tube (see Figure 5.7). The coarse mesh displays no cavitation underneath the hub nose cone, the medium mesh slight vapor in this region, while the fine mesh had the greatest amount of vapor beneath the hub. As this frequency is not excited when too little cavitation occurs beneath the hub it is thought to be excited by a critical amount of vapor in the draft tube. However, there is no definitive evidence at the moment to support this hypothesis, but, it will be discussed further in a qualitative manner when the complete results are presented. Other than the 7.33/rev fluctuations, the dominant frequencies of the various meshes are fairly similar. The time-step refinement shows that the dominant approximately 7.33/rev oscillations are present in all time-steps, as they all have larger amounts of vapor in the draft tube than the coarse and medium meshes.

The fine mesh case, with the smallest time-step, displays the most converged results from the time-averaged and unsteady refinement studies. Additionally, it provides the lowest $Co_{\text{max}}$ of all time-steps, which implies better accuracy in the regions with high $Co$, and also has an adequate $Co_{\text{mesh}}$. Finally, this case shows the most similar qualitative cavitation characteristics when compared with the experiments. Therefore, the fine mesh and time-step of $\Delta t = 0.0006$ s are employed.
for the remaining simulations in this chapter.

Figure 5.5: Convergence of normalized torque fluctuations on blade-1.
Figure 5.6: Convergence of normalized pressure fluctuations at probe-1.

(a) Peak-to-Peak Mesh Convergence
(b) Peak-to-Peak Temporal Convergence

(c) Spectral Mesh Convergence
(d) Spectral Temporal Convergence

Figure 5.7: Images of vapor content beneath the hub for various mesh levels.

(a) Coarse  (b) Medium  (c) Fine
5.3 Unsteady Results

Unsteady single and multiphase DES cases are presented and analyzed to understand the affect of cavitation on torque and pressure fluctuations at model and prototype scales. The DES simulations have both URANS regions and LES regions depending on the distance a cell is from the wall, and the cell volume. To display the distinct URANS and LES zones, the length $d - \Psi C_{DES} \Delta \geq 0$ is shown in Figure 5.8 for the representative model-scale Case 2 ($\sigma = 0.99$), with the finest mesh and smallest time-step. Any non-zero region (any region not the darkest shade of blue), is an LES region where $\mu_T$ is calculated with the SGS model. The regions of the darkest shade of blue (which are either 0 or less than zero) are URANS regions, which rely on the S-A model for the calculation of $\mu_T$. Thus, the URANS zones are isolated close to the walls, while the LES zones make up a large percentage of the vane/blade flow passage volume, as well as the central draft tube volume. Figure 5.8 shows the turbulent viscosity in the domain as well, and that LES acts to reduce $\mu_T$ in many highly turbulent separated flow regions (such as in the wake of the hub nose cone).

Figure 5.8: Contour plots of $d - \Psi C_{DES} \Delta \geq 0$ [m] (where DES regions are for values greater than zero), and eddy viscosity $\mu_T$, for a coordinate slice through the guide vanes, runner, and draft tube of the fine mesh and smallest $\Delta t$ in model-scale Case 2 ($\sigma = 0.99$).
5.3.1 Time-Averaged Unsteady

The steady performance of the machine is obtained by taking a time-average of the machine efficiency for each case. These efficiencies, displayed in Figure 5.9 are compared with those from the steady-periodic CFD results, the model tests, and the standard scalings for the prototype. As with the steady-periodic results, the unsteady results under-predict $\eta$ relative to the model experiments and prototype scaling. Even though the unsteady results contain the entire machine, and are completely aperiodic, the $\eta$ values are slightly more under-predicted than the steady-periodic results. However, the unsteady efficiencies are still fairly accurate, and they display the similar increase between model and prototype that is seen in the steady periodic results. Again, this increase in $\eta$ between scales is not as large as expected from the standard scaling, and this is thought to be due to the prototype running at a higher flow coefficient than the model. This increase in flow coefficient $\phi$ between the model and prototype is again seen in the unsteady results. The time-averaged $\phi$ for Cases 2 and 3 increases by roughly 2% which is similar to the increases seen in the steady-periodic results.

Figures 5.10 and 5.11 give a snapshot of the unsteady model cavitation on the guide vanes, runner blades, and beneath the nose cone in the draft tube, from the CFD and experiments of Case 2 ($\sigma = 0.99$). All of the cavitation structures that were seen in the experiments are captured by the CFD. The hub vortex cavitation is captured more accurately than the steady-periodic case, when compared with the experimental photos and drawings. Moreover, DES captures the cavitating precessing vortex core which was seen in the experiments. The model cavitating PVC displays fluctuations in vapor volume, multiple times per revolution. Furthermore, the unsteady CFD shows shedding and collapse of vapor structures on the suction-side of the runner blade, along with shedding and collapse of parts of the cavitating hub vortex, as it precesses in the draft tube. The unsteady features will be given much further attention in the following analysis. Figure 5.12 shows that Case 2, for the prototype, has slightly more cavitation in all of the same regions as the model. This is in agreement with the steady-periodic results and is caused by the lack of $Fr$ similarity between the scales. The prototype cavitating PVC displays oscillations in cavity volume as well, at multiple times per revolution, although not quite as strongly as in the model case.
Figures 5.13 and 5.14 show a snapshot of cavitation in model Case 3 ($\sigma = 0.99$) and the experiments, respectively. Also, Figure 5.15 displays cavitation in the prototype for Case 3. Clearly, the lower Thoma number cases have more vapor content beneath the overhang of the guide vane, on the leading-edge suction-side of the runner, and underneath the hub nose cone, as expected. The model-scale vapor compares well with the experimental images and drawings. A straight torch-like structure is visible below the nose cone, which becomes a precessing helical structure farther downstream (as it winds around a recirculation zone). At times, a small amount of vapor on the runner blade may shed from the surface sheet-like cavitation, convect downstream as a vapor cloud, and eventually collapse in a higher pressure zone.

The prototype has a similar vapor structure to the model beneath the hub, but shows more vapor content on the runner blade and shroud surface. This is in agreement with Case 3 of the steady periodic results. The signature of the vortices...
which shed from the runner leading-edge, and sometimes carry a vapor cloud with them, can be seen in the blade suction-side surface pressure field (on the blade in the center of Figure 5.15). They appear as equally spaced low pressure circles. Larger vapor clouds shed from the prototype blades than the runner blades.

**Runner Load Imbalance**

The unsteady results display a runner blade load imbalance as the blades rotate. In other words, the torque measured on a single-blade varies periodically with every revolution. Since the pressure just upstream of the runner proved to be fairly axisymmetric, while the pressure just downstream of the runner is not (see Figure
5.16, it is believed that the imbalance is caused by the draft tube. In fact, it is most likely due to the sharp elbow of the draft tube and the lack of any conical diffusion region beneath the runner (i.e. shallow draft tube). On the elbow side, much lower pressure can be seen than on the opposite side of the draft tube entrance.

This imbalance can prove even more problematic when cavitation occurs. For instance, greater amounts of vapor can form on the blades while in the low-pressure side of the machine. Then, since the blades carry along that vapor as they rotate, the vapor begins to collapse more violently when it enters the high-pressure side of the machine. The imbalance of vapor is shown in Figure 5.17. Finally, it is important to realize that the draft tube pressure imbalance is a steady phenomenon which is also seen in the steady-periodic results. The time-averaged difference between the pressure measured with probe-1 (high-pressure side) and probe-2 (low-pressure side) is 35% of $\rho_l E$. 

Figure 5.12: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for unsteady Case 2 ($\sigma = 0.99$).
Figure 5.13: Model-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for unsteady Case 3 ($\sigma = 0.70$).

Figure 5.14: Cavitation drawings and photo from Case 3 ($\sigma = 0.70$) model tests.
Figure 5.15: Prototype-scale guide vane, runner, and draft tube, displayed with isosurfaces of vapor volume fraction $\alpha_v = 0.2$ for unsteady Case 3 ($\sigma = 0.70$).

Figure 5.16: Piezometric pressure contours at the exit surface of the runner.
Figure 5.17: Prototype $\sigma = 0.99$ with vapor volume fraction $\alpha_v = 0.2$ at 3.1059 rev.
5.3.2 Precessing Vortex Core

The PVC is captured by the DES turbulence modeling for all of the cases. The helical structure is shown in Figure 5.18 for the single-phase model and prototype cases. The vortex is visualized with Q-criterion iso-surfaces of $100 \text{s}^{-1}$. Q-criterion is a method which has proven to be effective at bounding vorticies or coherent structures (more so than simply iso-surfaces of pressure). Originally an eddy was defined by Hunt et al. (1988) as a region with a positive second invariant of the velocity gradient tensor, where the pressure is also lower than the ambient value. Along with $p < p_{\text{amb}}$, Q-criterion $Q_{cr}$ can be expressed as

$$Q_{cr} = -\frac{1}{2} \nabla \mathbf{v} : \nabla \mathbf{v}^T.$$  

(5.3)

$Q_{cr}$ physically represents the local balance between shear strain rate and vorticity magnitude (Jeong and Hussain, 1995).

Precessing vortex cores, such as those shown in Figure 5.18, are uncommon at on-design conditions such as these. It usually occurs at part-load (Dörfler et al., 2013), however, its occurrence here may be due to the extremely poor performance of the draft tube. The prototype PVC displays a double-helix vortex at times, but, at most times it is a single-helix. In the cavitating results the vapor always forms in a single-helix in the prototype. The double-helix PVC is known to occur in a small range of flow conditions and no practical problems have been reported from it (Dörfler et al., 2013).

The pressure pulsations caused by the PVC are usually around one-third of a runner revolution (Dörfler et al., 2013), but a relation derived by Foroutan (2015), that seems to be in good agreement with experiments, shows that the ratio of angular frequency of PVC induced pressure pulsations in the draft tube $\omega_{pvc}$, to runner angular frequency, is a function of flow coefficient:

$$\frac{\omega_{pvc}}{\Omega} = \left(\frac{\pi^2}{8}\right) \phi.$$  

(5.4)

Thus, in this case, the pressure pulsations will occur at approximately 0.5/rev. Due to this very low frequency, many runner revolutions are required to adequately capture the correct magnitude and frequency of the pressure pulsations induced by the PVC. The results presented here are only for approximately 3 revolutions after
stationary operating conditions are reached. Thus, less than two PVC induced pulsations are captured. The frequency analysis will show that a sub-revolution pressure pulsation is captured in the draft tube, although it appears that the PVC is precessing at roughly 1/rev in the flow visualization. However, there are two reasons why frequencies this low will not be investigated deeply in this research. First, many more runner revolutions are necessary to make sure the sub-revolution oscillations have truly converged to a stationary state and to correctly determine the amplitude and frequency of the pulsations. Secondly, with the smallest time-step, 3 revolutions does not provide enough frequency resolution (a larger sample size is needed) to accurately pin down the amplitude or frequency of the sub-revolution pulsations. Therefore, these low frequencies will be acknowledged but not discussed in much detail, leaving the focus of the analysis on oscillations which occur between 1 and 60 times per revolution.

5.3.3 Unsteady Torque on Runner Blades

The unsteady torque on the runner can be investigated through the time-series data and amplitude spectra of a single blade or for the entire runner. For the single-blade cases, the time-series of blade-1, for the model/prototype single-phase, $\sigma = 0.99$, and $\sigma = 0.70$ cases are shown in Figure 5.19.
Figure 5.19: Time-series of normalized torque fluctuations for blade-1.
Table 5.2: Peak-to-peak amplitudes of normalized torque readings for blade-1.

<table>
<thead>
<tr>
<th>Case</th>
<th>Scale</th>
<th>$T_{E p-p}^{\text{blade-1}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>[#]</td>
<td>[-]</td>
<td>[%]</td>
</tr>
<tr>
<td>1-phase</td>
<td>Model</td>
<td>0.78</td>
</tr>
<tr>
<td>1-phase</td>
<td>Prototype</td>
<td>1.29</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Model</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Prototype</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Model</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Prototype</td>
</tr>
</tbody>
</table>

The clearly dominant fluctuations for all of the cases are the $1/\text{rev}$ oscillations. These fluctuations are due to the previously discussed pressure imbalance at the draft tube inlet, caused by the elbow in the shallow draft tube. The torque is higher than the mean when the blade is passing through the low-pressure region (closer to the elbow), and lower than the mean as it passes through the high-pressure region. Roughly, a 2% difference in torque, relative to the net torque on all blades, is seen on a blade from one side of the turbine to the opposite side. Relative to the mean torque on a single-blade, this is a 22% difference. This dominant frequency is modulated by higher frequencies.

As a blade rotates into the high-pressure region, vapor formed in the low-pressure region has a higher chance to collapse and produce fairly impulsive, stochastic spikes in the torque that do not occur in the single-phase results. These spikes are more prominent in the prototype torque (when compared with the model torque) where larger amounts of vapor are violently collapsing on the suction-side of the runner blade. Additionally, as Thoma number is reduced, and more cavitation occurs in the turbine, the impulsive spikes in the torque grow larger and more frequent. The peak-to-peak amplitudes of the torque on blade-1 are shown in Table 5.2. In both the single-phase and cavitating results, a broad sharp decrease in torque occurs near the end of the low-part of the torque oscillations. This decrease is due to a circumferentially localized region of the highest pressure at the draft tube inlet.

The amplitude spectra of blade-1 is shown in Figure 5.20. The dominant frequency is clearly the $1/\text{rev}$ oscillation from the pressure imbalance in the draft tube. All cases (single-phase model/prototype and cavitating model/prototype) display similar spectra. The amplitudes of the dominant fluctuations, and the
20/rev components, are roughly the same. The 20/rev oscillation is the higher frequency component which was seen to be modulating the 1/rev fluctuation of torque in the time-series. It is due to the blade-vane interaction, as a single runner blade passes 20 guide vanes in one revolution. Harmonics of this interaction are seen at 40 and 60/rev. Also, a small 3/rev oscillation is visible for each case, which may be a harmonic of the 1/rev.

The only visible difference in the amplitude spectra between the single-phase and the cavitating cases arises through the higher broadband content of the cavitating cases. The impulsive torque spikes, caused by cavitation collapse, produce this broadband signal. The broadband content is higher for the prototype cavitating flow than for the model, as larger magnitudes and number of spikes were seen in its time series relative to the model. Moreover, the broadband signal increases as cavitation occurs to a greater extent, as evidenced by the differences between Case 2 and Case 3.

Visualization of the vapor cavity collapse on the runner blade surface can be
Figure 5.21: Image series depicting vapor collapse on prototype runner blade surface for Case 2 ($\sigma = 0.99$).

seen in Figure 5.21. This series of images depicts the prototype runner ($\sigma = 0.99$) passing through the wake of a guide vane in the high-pressure region of the draft tube. As the runner approaches the wake, the vapor on the leading-edge suction side of the runner begins to shrink. Then it violently collapses and produces an extremely localized (temporally and spatially) high-pressure on the runner surface. A similar phenomenon occurs in the model turbine when $\sigma = 0.99$, although there is less vapor present. While this high-pressure may not have much affect on the torque on a single blade (other than to raise the broadband content slightly) it could cause erosion and damage to the blade structure. A more detailed analysis concerning the maximum pressure impulses on the blade surface, and commentary on the potential damage due to them, can be found in Section 5.3.5.
As the Thoma number is decreased, more cavitation occurs on the blades, shroud, and beneath the guide vanes. When $\sigma = 0.70$, the vapor on the prototype shroud coalesces with the leading-edge suction-side runner blade vapor as it passes (see Figure 5.22). Then, the larger cavity sheds from the blade as it passes through the guide vane wake. The vapor cloud travels above the blade surface, and through the blade passage, until it collapses in a higher pressure region. This cycle then repeats as the runner passes by the shroud vapor again. Additionally, vortices shedding from the leading-edge of the runner, sometimes carrying along small vapor clouds with them, which collapse above the surface as they convect. Similar mechanisms occur in the model case at this Thoma number, but less vapor is seen.

Another unsteady torque which can be examined is that of the net torque on
the entire runner (shaft torque). Figure 5.23 depicts the amplitude spectra of the net torque on the entire runner. Immediately, it is evident that a 1/rev oscillation is much more dominant in the single-phase cases than the cavitating cases. Also, the single-phase and cavitating prototypes are more dominant at 1/rev than the corresponding model cases. The damping of typical machine oscillations between single-phase and cavitating cases is well documented in the literature (Guo et al., 2007; Panov et al., 2014; Wang and Chang, 2010). This 1/rev fluctuation is the machine frequency and is caused by the same mechanism that causes the 1/rev fluctuation in the single-blade.

It is apparent that the runner torque does not experience the same 20/rev oscillations that a single blade does, but rather a 22/rev peak appears. Note that the interaction of the runner and guide vanes occurs 220 times per revolution (20 guide vanes and 11 runner blades) when observing the shaft torque. However, little content appears at 220/rev (not shown). Thus, the effects of blade-vane interaction are absent from the net runner torque, and this is not surprising, as the runner
is designed to avoid issues with blade-vane interaction on the net torque. The 22/rev peaks are thought to be related to the pressure imbalance at the runner exit. As each blade will enter the high-pressure side, and enter the low-pressure side, once every revolution, this can result in 22/rev fluctuations on the entire runner. All cases display an 11/rev peak which is equivalent to the number of blades in the machine and thought to be related to a phenomenon seen in the single-blade torque. That is, the 11/rev peak may be attributed to the 1/rev torque decreases which occur as each runner passes the localized high-pressure region in the draft tube. When these are summed over all blades, they do not interfere with each other, but rather produce 11 decreases per revolution. This occurrence may also be accentuated by the fact that it is a sub-harmonic of 22/rev (and vice-versa). Cavitation can produce unique oscillations which would not appear in single-phase flow. Such a unique fluctuation arises at 7.33/rev (a sub-harmonic of 22/rev). This oscillation only occurs in the $\sigma = 0.99$ case, and is more prominent in the model. It will be discussed further in Section 5.3.4.

While the 11 and 22/rev frequencies do not display large differences between the single-phase and $\sigma = 0.99$ results, for $\sigma = 0.70$ a large increase in the 22/rev fluctuations is visible. They become the dominant frequency for that Thoma number. This is may be due to more violent vapor collapse, and growth, as a blade passes into and out of the high-pressure side. Notice that the model-scale has a larger amplitude here but the prototype-scale has much more broadband content. The aperiodic pressure spikes give rise to the increase in the broadband nature of the amplitude spectra for the cavitation cases. The differences in broadband content between the various cases can be viewed better on a log$_{10}$-scale in Figure 5.24. Clearly, the broadband nature increases as more cavitation occurs. This can lead to problematic structural vibrations in the runner, as the natural frequencies of the structures are more likely to be excited.
5.3.4 Pressure Fluctuations

To reiterate that which was already stated, the pressure probes for which data is presented are located on the high-pressure side near the draft tube inlet (probe-1), the low-pressure side near the draft tube inlet (probe-2), and far upstream in both sections of the penstock (probe-A and probe-B). Figure 5.25 displays the differences in the time-series’ of pressure probe-1 in the model and prototype for single-phase flow and cavitating flow (Case 2: $\sigma = 0.99$ and Case 3: $\sigma = 0.70$). It is quite obvious that the cavitating cases experience much larger peak-to-peak pressure fluctuations at probe-1, and contain frequencies which do not seem to appear in the single-phase cases. The peak-to-peak amplitudes of normalized pressure for all of the probes are shown in Table 5.3. Besides probe-2, which is possibly more affected by the draft tube vortex, the peak-to-peak amplitudes from all the pressure probes are higher in the cavitating cases.
Figure 5.25: Time-series of normalized pressure fluctuations at probe-1.

(a) Model 1-phase  
(b) Prototype 1-phase

(c) Model $\sigma = 0.99$  
(d) Prototype $\sigma = 0.99$

(e) Model $\sigma = 0.70$  
(f) Prototype $\sigma = 0.70$
Table 5.3: Peak-to-peak amplitudes of normalized pressure readings for all probes.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1-phase</td>
<td>0.78</td>
<td>Model</td>
<td>0.78</td>
<td>3.64</td>
<td>0.16</td>
<td>0.17</td>
</tr>
<tr>
<td>1-phase</td>
<td>1.29</td>
<td>Prototype</td>
<td>1.29</td>
<td>4.64</td>
<td>0.18</td>
<td>0.21</td>
</tr>
<tr>
<td>2</td>
<td>0.99</td>
<td>Model</td>
<td>3.93</td>
<td>4.19</td>
<td>1.27</td>
<td>1.51</td>
</tr>
<tr>
<td>2</td>
<td>3.00</td>
<td>Prototype</td>
<td>3.00</td>
<td>3.93</td>
<td>0.92</td>
<td>1.16</td>
</tr>
<tr>
<td>3</td>
<td>0.70</td>
<td>Model</td>
<td>2.97</td>
<td>3.50</td>
<td>0.65</td>
<td>0.87</td>
</tr>
<tr>
<td>3</td>
<td>5.90</td>
<td>Prototype</td>
<td>5.90</td>
<td>5.71</td>
<td>0.81</td>
<td>1.05</td>
</tr>
</tbody>
</table>

By examining the amplitude spectra of probes 1, 2, A, and B in Figure 5.26, it is seen that the $\sigma = 0.99$ case displays unique pressure pulsation behavior that is more prominent in the model. In these cases, probe-1 is dominated by a 7.33/rev oscillation which does not occur in the other simulations. As expected from the time-series analysis, the model peak is much higher than the prototype peak. Furthermore, all cases display a sub-1/rev fluctuation in probe-1, thought to be caused by the PVC, which is also visible in the time series. This oscillation is more prominent in the spectra of the cavitating cases.

Probe-2 displays similar features to probe-1. However, probe-2, being on the elbow-side of the draft tube and possibly having stronger interaction with the PVC (Dörfler et al., 2013), displays much more content at 1/rev and sub-1/rev than probe-1. In fact, these fluctuations are dominant in this probe. Probe-A and probe-B are fairly similar to each other. The only visible fluctuations occur in the cavitating cases, and most noticeably in model $\sigma = 0.99$ case. Thus, far upstream of the runner, the lowest frequencies have no significance, and the spectra are dominated by the 7.33/rev pulsations. In all probes, the broadband content is visibly higher in the cavitating cases due to the impulsive pressure spikes from vapor collapse. To accentuate this fact, for reasons concerning structural vibration induced by cavitation collapse, the pressure spectra for probe-1 are plotted on a $\log_{10}$-scale in Figure 5.27.
Figure 5.26: Amplitude spectra of normalized pressure fluctuations.
Figure 5.27: Amplitude spectra of normalized pressure fluctuations at probe-1 scaled by log_{10}.  
The cause of the 7.33/rev content for the $\sigma = 0.99$ case is difficult to determine, but certain traits are known:

1. This mechanism is only present when cavitation is present.

2. It occurs throughout the machine, including in the penstock, at the draft tube exit, and appears in fluctuations of the mass flow rate at the inlet to the machine.

3. It only seems to occur when vapor occurs to a certain extent beneath the hub (recall the coarse and medium meshes from the mesh convergence study).

4. It appears in the fluctuations of the total vapor volume fraction in the machine (see Figure 5.28).

5. It is more dominant in the model than the prototype.

6. The frequency seems to link up with a visual inspection of the oscillations in the vapor beneath the hub nose cone. At points, the fluctuations cause shedding and collapse of vapor (see Figure 5.29).

It is not certain whether the vapor fluctuations of the cavitating hub vortex are the cause or an effect of the pressure pulsations through the machine. It may be reasonable to think that cavitation in the draft tube allows the draft tube frequency itself to be excited. This is indeed a common occurrence (IEC, 1999), and is the reason for many of the draft tube surge studies investigated in Chapter 1. Additionally, if the vapor is modifying the natural frequency of the flow conduit to 7.33/rev, then it could easily be excited as a sub-harmonic of the 22/rev fluctuations. Regardless of the source, these pulsations are unique to $\sigma = 0.99$ cases, produce flow conditions which persist throughout the entire machine, and the simulations have proven effective at capturing these traits.
Figure 5.28: Amplitude spectra of normalized volume of vapor fluctuations.

Figure 5.29: Image series depicting partial shedding and collapse of hub vortex vapor in model Case 2 ($\sigma = 0.99$).
5.3.5 Maximum Pressure on Runner Blade Surface

While the torque measurements display the evidence of cavitation collapse on the blade surfaces, the integrated force quantities mask the true amplitude of these highly localized events. To allow for an analysis of the amplitude of the pressure spikes that occur on the runner blade surface, due to vapor collapse, and to provide quantitative data for qualitative commentary on potential cavitation damage, the maximum pressure on blade-1 is monitored at every time-step. The maximum pressure is monitored because the spatial location of cavitation collapse is not known \( a \text{ priori} \), and moreover, this location is not consistent, as it is extremely localized and the vapor may collapse differently at each occurrence.

Figure 5.30 displays the maximum pressure time-series of the model and prototype for single-phase and cavitating flow. It is immediately evident that large pressure spikes manifest in the cavitating cases which do not occur in the single-phase flow. For the \( \sigma = 0.99 \) case, the model spikes can reach up to twice the net pressure difference across the machine, while the prototype collapse is more violent, reaching almost 3 times the net pressure difference. In the \( \sigma = 0.70 \) case, the model spikes can reach up to 6 times the net pressure difference across the machine, while the prototype collapse is again more violent, reaching over 7 times the net pressure difference. For the \( \sigma = 0.99 \) case, vapor collapse only seems to occur as the runner blade enters the high-pressure side of the machine, but in the \( \sigma = 0.70 \) case, collapse happens even when the blade is entering the low-pressure side of the machine. The pressure spikes are fairly aperiodic, but they seem more likely to occur just as the blades begin to pass through the wake of the guide vanes. The 20/rev pulsation is dominant in these maximum blade pressure plots (see Figure 5.31) for both single-phase and cavitating flow.
Figure 5.30: Time-series of normalized maximum pressure on blade-1.
The oscillations of the single-phase flow primarily follow the high-pressure at the leading-edge of the blade, close to the shroud. As the blade passes through the wake of the guide vane it first experiences a pressure increase above the mean, followed by a pressure decrease below the mean. This is similar to the force exerted on a surface interacting with a vortex (Howe, 2003). Thus, the 20/rev pulsation is due to the blade-vane interaction. A larger, broader pressure spike occurs as the blade passes by the localized high-pressure zone on the high-pressure side of the draft tube. Furthermore, the 1/rev oscillation is evident, caused by the pressure imbalance at the entrance to the draft tube. It is also apparent the cavitation beneath the guide vane overhang reduces the magnitude of the runner-vane interaction (at least when vapor is not collapsing on the blade surface). This effect occurs because the pressure differences across the wakes are reduced due to the fact that the pressure of the cavity remains close to the vapor pressure. The single-phase flow is not constrained in that manner. Since vapor collapses on the surface for the $\sigma = 0.70$ case, whenever the blade passes through the guide vane wake, these cases display the
largest peaks at 20 and 40/rev. Finally, a broadband nature arises in the amplitude spectra for the single-phase cases (thus, not just due to the pressure impulses from vapor collapse), which is accredited to the maximum pressure reading, as it is not constrained to a particular location.

5.4 Commentary on Potential Cavitation Erosion

Erosion to the blade surface based on these simulations is difficult to predict quantitatively, especially for such a massive case. One of the leading theories as to how cavitation generates high enough pressures to pit steel surfaces, is the micro-bubble/micro-jet argument (Dular and Coutier-Delgosha, 2009). It is believed that micro-bubbles are present throughout the liquid, and can be on or close to the blade surface. Then it is thought that large-amplitude pressure impulses, formed by large-scale cavitation collapse, impinge on the surface and the micro-bubbles. The interaction of the pressure wave with the micro-bubbles, deforms the bubble into a toroidal shape sending a high velocity liquid micro-jet into the blade surface. This high velocity liquid jet is thought to be the source of the pitting, while the large-scale cavitation collapse is the critical parameter affecting the micro-jet formation and speed. The physical process is displayed in Figure 5.32. The pits build through an incubation period of many cavitation events, until the surface becomes rougher and more jagged, and eventually the micro-jet impacts begin to erode away material from the surface.

Dular and Coutier-Delgosha (2009) mention that large-scale vapor cloud collapse can cause pressure waves reaching several MPa. While this is 2 orders of magnitude short of the yield strength of a runner blade, it is only the driver of the micro-bubble/jet mechanism, and doesn’t directly pit the surface. The micro-jet duration
is roughly 1 ns and affects an area of several micrometers, but can reach velocities of several hundred m/s and cause pressures of larger than 1 GPa (Dular and Coutier-Delgosha, 2009), which is much larger than the yield strength of most metals.

At present, the temporal and spatial scales of the micro-jet mechanisms are much too small to resolve for large-scale industrial problems. However, the pressure spikes caused by large-scale vapor collapse are resolved, to a certain degree, in these results. To allow for accurate capturing of pressure spikes, the compressibility of multiphase flow must be taken into consideration (Dular and Coutier-Delgosha, 2009). Then it will be possible to insert these pressures into a cavitation damage model that can predict the micro-jet velocity and the damage to the blade (Dular and Coutier-Delgosha, 2009; Dular et al., 2006). Accurate results will also require strong coupling with experiments. This is the subject of future work in this area.

Without considering the entire damage model here, some basic calculations can be made, in an attempt to understand what the pressure spikes in the cavitation simulations imply in terms of damage to the blade surface. Based on the work of Plesset and Chapman (1971), the velocity of the jet is defined as

\[ v_{jet} = 8.97\xi^2 \sqrt{\frac{p_{abs} - p_v}{\rho_l}} \]  

(5.5)

where \( \xi \) is the non-dimensional distance from the spherical micro-bubble center to the surface, and the pressure applied to the surface by the impacting jet (water hammer) can be approximated as

\[ p_{wh} \approx v_{jet} \rho_l c_l \]  

(5.6)

where \( c_l \) is the speed of sound in the liquid. Using the maximum pressure from the large-scale cavitation collapse in prototype Case 3 (\( \sigma = 0.70 \)), seen in Figure 5.30, as input, and \( \xi = 1.1 \) for a micro-bubble close to the surface as in Dular and Coutier-Delgosha (2009), \( v_{jet} \) is predicted to be 329 m/s. The impact pressure of the jet on the runner blade surface is 490 MPa. This is greater than the yield strength of antiquated carbon steel runner blades (\( \approx 300 \) MPa), and around the yield strength of many of the stainless steel runner blades installed in state-of-the-art hydroturbines (\( \approx 500 \) MPa). In contrast, the maximum pressure spikes in prototype
Case 2 ($\sigma = 0.99$) result in a micro-jet impact pressure of 345 MPa. Moreover, the simulations are most likely under-predicting the magnitude of the surface pressure from large-scale vapor collapse. To improve this prediction, more mesh and temporal resolution are necessary, but additionally, compressible (i.e. variable phasic densities) flow must be considered, to capture shock waves in the liquid-vapor mixture. Even if the yield strength of the material is not reached, and no immediate damage occurs to the blade, damage can still accumulate with repetitive collapse events at the same location (cavitation fatigue damage). When $p_{wh}$ exceeds the yield strength of the material, where the material is assumed to behave rigidly in the model unless that threshold is surpassed, Dular and Coutier-Delgosha (2009) show how the depth and radius of a pit can be determined, and how the damage can be extrapolated in time. Note that this model is only for the incubation period of cavitation damage, where plastic deformation occurs on the surface, but before mass loss occurs to the material.

Nevertheless, it can be noted that, the greater the maximum pressures spikes from large-scale cavitation collapse, shown in Figure 5.30, the higher the probability of damage to the runner surface at that location. Thus, cavitation damage on the runner blade surface is most likely to occur on the leading-edge of the suction side of the blade, by the shroud, as the blade begins to pass through the wake of the guide vane on the high-pressure side of the machine. Furthermore, damage is more likely to occur in the prototype than the model for the cases shown. Additionally, when Thoma number is lowered a slight amount, the pressure spikes increase by a great deal, even though the steady-state performance of the machine is hardly affected. This increases the chance of cavitation-induced damage to the runner blades in the lower Thoma number case, and highlights the potential benefits that unsteady multiphase CFD can deliver to the hydroturbine industry.

### 5.5 Summary

The unsteady results displayed time-averaged performance behavior that compared favorably with the steady-periodic results and the experiments. No unsteady experimental data was available for direct quantitative comparison between the torque and pressure fluctuations in the machine, but, the cavitation structures observed in the simulations were qualitatively consistent with those surveyed in
the experiments. These structures included the precessing vortex core in the draft tube.

Several novel contributions were made by the simulations presented in this chapter. The cavitating simulations conducted were the first of DES-type for the entire machine, as previously only certain cavitating components were studied in isolation with LES/DES. As with the steady-periodic results, unsteady cavitating simulations of both the model and prototype, and their comparison, were distinct contributions to the field. For the first time, shallow draft tube induced runner imbalance was shown to have a direct effect on the asymmetric distribution of vapor about the runner. Moreover, this phenomenon was shown to contribute to fluctuations in the torque on the blades at certain circumferential positions. These fluctuations were shown to be due to cavitation collapse, and were partly driven by the imbalance. Another contribution was the appearance of unique frequency components, only caused when vapor appeared in the draft tube, which persisted throughout the entire machine. Lastly, the maximum pressures on the hydroturbine blades were monitored, which displayed pressure spikes caused by cavitation collapse, and allowed for commentary on potential erosion.
Summary and Conclusions

6.1 Overview

High-fidelity multiphase CFD simulations of model and prototype scale Francis hydroturbines were conducted to expand the state-of-the-art in the computation of cavitating flows in hydroturbines and to provide unique analyses of the machine performance and fluid dynamics. A clear and concise literature review was provided, leading to the value of the contributions presented in this dissertation. The numerical methods and simulation approaches were provided in detail for cavitating steady-periodic simulations and unsteady cavitating computations of the entire machine. Detailed mesh and time-step convergence studies were conducted to ensure convergence of the numerical results.

Steady-periodic results compared favorably with experimental data, and were used to analyze the performance breakdown of the machine as cavitation became more extensive in the turbine. Moreover, these results were extended beyond the experimental cases, and critical performance differences between model and prototype scales were reported, alongside visible cavitation content disparities between the scales. A stage-by-stage performance breakdown of the model and prototype machines was conducted to show the affect of cavitation on the performance of each component of the machine, separately.

Time-averaged results from the unsteady DES of the entire machine displayed qualitative agreement with the experimental and steady-periodic results. As there was no experimental data available for the unsteady fluctuations in the machine, this was the only possible quantitative comparison that could be made between the simulations and reality. However, the cavitation structures observed in the computational results were qualitatively consistent with those observed in the
experiments. Unsteady torques and pressures were analyzed for single-phase and
cavitating model and prototype turbines, and the differences between the cases were
highlighted. Key aspects of the flow conditions which influenced the problematic
features of unsteady cavitating flow were unveiled and discussed. Finally, a short
commentary was given on the role state-of-the-art large-scale cavitation simulations
can provide to erosion and wear prediction in industrial applications such as this.

6.2 Contributions and Findings

The following unique contributions which were made to the state-of-the-art in
cavitating hydroturbine CFD are listed below:

• Comparison of model and prototype cavitating CFD results (steady-periodic
  and full-unsteady) to reveal effects from lack of $Re$ and $Fr$ similitude

• Stage-by-stage performance decomposition of steady-periodic results

• DES of fully-coupled cavitating hydroturbine

• Simulation and analysis of unsteady cavitating runner and runner-vane inter-
  action.

• Quantitative analysis of pressure impulses on runner blades due to vapor
  collapse and qualitative analysis of potential for runner blade damage.

The following original findings of this research are provided below:

• Steady-state cavitation performance breakdown occurs much more abruptly
  in the model, as the larger hydrostatic pressure gradient in the prototype
  helps prevent vapor formation in the draft tube.

• As cavitation breakdown occurs, the losses in the machine occur primarily in
  the draft tube and increase in the draft tube much more abruptly than the
  other stages.

• Losses in the model and prototype runners and guide vanes converge as
cavitation occurs to a greater extent, while the difference in the draft tube
  losses between the two scales increases.
• Shallow draft tube induced runner imbalance causes an asymmetric vapor distribution about the runner.

• Vapor is more likely to form as the blades rotate into the low-pressure side of the machine, and more likely to collapse (producing large pressure spikes) as the blades rotate into the high-pressure side of the machine.

• Unique frequency components appear and persist throughout the entire machine only when cavitation is present at the hub vortex to a certain degree.

• Large cavitation-induced maximum pressure spikes on the runner blade surfaces, may be able to provide input to future modeling analysis of cavitation erosion to the runner blades. The largest maximum pressures were around values of yield strength for modern hydroturbine runners.

• As Thoma number is reduced, the pressure spikes increase in amplitude by a great deal, even when the steady-state performance of the machine does not change much.

6.3 Future Work

There are many logical extensions to the results presented in this dissertation. For the steady computations, the next step is to compute a numerical hill chart from steady-periodic simulations at various guide vane opening angles. Thus, instead of simulating the common cavitation breakdown test, all operating conditions will be computed with multiphase CFD and vapor will only appear if local pressures drop below the vapor pressure. This will provide the first true numerical hill chart to the literature, as other attempts could not include cavitation effects, as they were only single-phase. As the steady-periodic method reduces computational effort, resources are certainly available to conduct this study right away.

A stage-by-stage performance decomposition of each operating condition on the numerical hill chart could follow. Guide vane, runner, and draft tube losses could be analyzed and compared with single-phase results. If the hill chart is computed for the model and prototype scales the variations due to lack of Reynolds and Froude similitude will clearly stand out, especially in the far off-design cavitating conditions.
For the unsteady computations, a logical next step is to simulate the current configuration for many more revolutions. This will allow for a more detailed investigation of the low frequency draft tube dynamics, which is a research area that has received a lot of interest lately. Another immediately realizable study, is to simulate the current configuration at the lower Thoma numbers presented in the steady-periodic computations. It would be particularly interesting to see the differences in the unsteady dynamics between the model-scale case with severe cavitation on the draft tube walls, and the corresponding prototype-scale case, which does not develop vapor on the draft tube walls. Furthermore, it would be interesting to study the unsteady cavitation field at various guide vane opening angles (part and overload conditions).

One of the most promising multiphase CFD studies could be into large-scale cavitation induced damage. To accomplish this, the computations presented here would require more temporal/spatial resolution, and compressible flow coupled with incubation-period damage models. However, the computational resources and numerical methods are already available to execute this study. Cavitation damage models, discussed in this dissertation, have the ability to display accurate spatial distribution and extent of damage, with accurate input from large-scale multiphase CFD. To be successful, it is of vital importance that this type of research be conducted concurrently with a well funded experimental investigation.

6.4 Conclusion

The results presented herein display the efficacy of capturing important hydroturbine cavitation characteristics with multiphase CFD. Multiphase CFD allows for the accurate analysis and comparison to hydroturbine cavitation experiments, and ease of extension to more severe cavitation breakdown cases and prototype-scales (which are more difficult to analyze experimentally). Thus, it is recommended as a tool which can be a beneficial supplement to model experiments. Besides the obvious industrial applications, cavitating hydroturbine CFD has the potential to resolve long standing research questions in the field, as well as uncover physical phenomena unique to hydraulic turbines. For this reason, many research avenues remain available for investigation.
A

Spalart-Allmaras DDES Model

The full details of the Spalart-Allmaras (S-A) Delayed Detached Eddy Simulation (DDES) turbulence model are readily available in the STAR-CCM+ User Guide (CD-adapco, 2014) (a concise description is given in Section 4.2.4), but are also included here for completeness. The S-A model solves a transport equation, shown in (A.1) in integral form, for the ‘modified diffusivity’ $\tilde{\nu}$.

$$\frac{\partial}{\partial t} \int_V \rho \tilde{\nu} dV + \int_S \rho \tilde{\nu} (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS =$$

$$\frac{1 + C_{b2}}{\tilde{\nu}} \int_S (\mu + \rho \tilde{\nu}) \nabla \tilde{\nu} \cdot \mathbf{n} dS - \frac{C_{b2}}{\tilde{\nu}} \int_V (\mu + \rho \tilde{\nu}) \nabla \tilde{\nu}^2 dV + \int_V \mathcal{G}_\tilde{\nu} - \Upsilon_{\tilde{\nu}} dV \quad (A.1)$$

In Equation (A.1), the terms $\mathcal{G}_\tilde{\nu}$ and $\Upsilon_{\tilde{\nu}}$ represent the production and dissipation of turbulence, while the first two terms on the RHS are the combined conservative and non-conservative diffusions. $\mathcal{G}_\tilde{\nu}$ is given as

$$\mathcal{G}_\tilde{\nu} = (1 - f_{t2}) C_{b1} f_{r1} \rho \tilde{S} \tilde{\nu} \quad (A.2)$$

where the rotation function $f_{r1}$ is

$$f_{r1} = (1 + C_{r1}) \frac{2r^*}{1 + r^*} \left[ 1 - C_{r3} \tan^{-1} (C_{r2} r^*) \right] - C_{r1} \quad (A.3)$$

$$r^* = \frac{\sqrt{2 \mathbf{S} : \mathbf{S}}}{\sqrt{2 \mathbf{W} : \mathbf{W}^T}} \quad (A.4)$$

$$\mathbf{S} = \frac{1}{2} (\nabla \mathbf{v} + \nabla \mathbf{v}^T) \quad (A.5)$$

$$\mathbf{W} = \frac{1}{2} (\nabla \mathbf{v} - \nabla \mathbf{v}^T) \quad (A.6)$$
\[ r^* = 2W_{ik} S_{jk} \left[ \frac{DS_{ij}}{Dt} \right] \frac{1}{D^4} \]  
\[ D^2 = \frac{1}{2} \left( 2S : S + 2W : W^T \right) . \]  
\[ (A.7) \]

The constants that appear in these equations are
\[ C_{r1} = 1 \quad C_{r2} = 12 \quad C_{r3} = 1. \]  
\[ (A.9) \]

Additionally,
\[ f_{t2} = C_{t3} \exp \left[ -C_{t4} \chi_{\tilde{\nu}}^2 \right] \]  
\[ (A.10) \]

where
\[ \chi_{\tilde{\nu}} = \frac{\tilde{\nu}}{\nu} \]  
\[ (A.11) \]

and the deformation parameter \( \tilde{S} \) is given as
\[ \tilde{S} = f_{\nu3} \sqrt{2W : W^T} + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{\nu2} \]  
\[ (A.12) \]

with
\[ f_{\nu2} = \left( 1 + \frac{\chi_{\tilde{\nu}}}{C_{\nu2}} \right)^{-3} \]  
\[ (A.13) \]
\[ f_{\nu3} = \frac{(1 + \chi_{\tilde{\nu}} f_{\nu1})(1 - f_{\nu2})}{\chi_{\tilde{\nu}}} \]  
\[ (A.14) \]
\[ f_{\nu1} = \frac{\chi_{\tilde{\nu}}^3}{\chi_{\tilde{\nu}}^3 + C_{\nu1}} \]  
\[ (A.15) \]

The turbulence dissipation \( \Upsilon_{\tilde{\nu}} \) is written as
\[ \Upsilon_{\tilde{\nu}} = \rho \left( C_{w1} f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right) \left( \frac{\tilde{\nu}}{d} \right)^2 \]  
\[ (A.16) \]

where
\[ f_w = g_{\nu} \left[ \frac{1 + C_{w3}}{g_{\nu}^6 + C_{w3}^6} \right] \frac{1}{\beta} \]  
\[ (A.17) \]
\[ g_{\nu} = r^\circ + C_{w2} \left( r^\circ 6 - r^\circ \right) \]  
\[ (A.18) \]
\[ r^\circ = \frac{\tilde{\nu}}{S_{K^2 d^2}} \]  
\[ (A.19) \]
The model coefficients are

\[
\begin{align*}
C_{b1} &= 0.1335 & C_{b2} &= 0.622 & \varsigma_{\tilde{\nu}} &= 2/3 \\
C_{\nu1} &= 7.1 & C_{\nu2} &= 5 & C_{w1} &= C_{b1} + \frac{1 + C_{b2}}{\varsigma_{\tilde{\nu}}} \\
C_{w2} &= 0.3 & C_{w3} &= 2.0 & \kappa &= 0.41 \\
C_{DES} &= 0.65 & C_{t3} &= 1.1 & C_{t4} &= 2.0
\end{align*}
\]  

(A.20)

For a RANS calculation, Equation (A.1) is solved for \( \tilde{\nu} \) and the eddy viscosity can be computed with

\[
\mu_T = \rho \tilde{\nu} f_{\nu1} 
\]  

(A.21)

where \( f_{\nu1} \) is a function of \( \tilde{\nu} \) and the kinematic viscosity \( \nu \). Thus, the S-A model provides the eddy viscosity for a RANS computation and can also be used with wall functions.

To compute the large eddies only, as is done in LES, the filtered Navier-Stokes equations must be used. Here, the velocity is filtered by a localized kernel function that has a length scale \( \Delta \) associated with it (Ferziger and Perić, 1996). If eddies are larger than \( \Delta \), then they will be resolved and do not need to be modeled. \( \Delta \) is the largest distance between the computational cell center under consideration and the cell center of any of its neighboring cells. If eddies are smaller than \( \Delta \) then they must be modeled by a sub-grid scale (SGS) model. The sub-grid scale terms can be represented as a hypothetical stress, in a similar manner to the RANS turbulence models. The SGS model used in this work is that of Smagorinsky (1963). The SGS eddy viscosity is represented as

\[
\mu_T = \rho \Delta^2 S, 
\]  

(A.22)

where \( S = \sqrt{2} \mathbf{S} : \mathbf{S} \) and \( \mathbf{S} = \frac{1}{2} \left( \nabla \mathbf{v} + \nabla \mathbf{v}^T \right) \). DES links RANS and LES together by solving the momentum equation (4.3) using the eddy viscosity determined from (A.21) (obtained by solving (A.1)) in RANS regions, and using the eddy viscosity determined from (A.22) in LES regions. This feat is accomplished through the turbulent length scale \( \tilde{d} \) (given below), that modifies the wall distance \( d \) and determines whether the mesh (and flow in the case of DDES) dictate a region as RANS or LES.

\[
\tilde{d} = d - f_{dmax} (0, d - \Psi C_{DES} \Delta) 
\]  

(A.23)
where $C_{DES}$ is a parameter capable of tuning DES. $C_{DES}$ is commonly set to 0.65, where simulations have proven to be insensitive to this parameter (Strelets, 2001). $\tilde{d}$ dictates whether the eddy viscosity will be based on the Reynolds stresses (A.21) or the SGS Reynolds stresses (A.22). In this dissertation, the DDES formulation is used to improve the ability of conventional DES to differentiate between RANS and LES regions (Spalart et al., 2006). DDES uses the function $f_d$ to accomplish this, and it is defined as

$$f_d = 1 - \tanh \left( \frac{(8r_d)^3}{3} \right)$$

$$r_d = \frac{\nu}{\sqrt{\nabla v \cdot \nabla v} \kappa^2 d^2}$$

(A.24)

DDES allows $\tilde{d}$ to depend on eddy viscosity and the velocity gradients, and thus, the flow itself, when determining whether a region of the mesh should use RANS or LES. The term $\Psi$ is a low-$Re$ correction function that prevents activation of low-$Re$ terms when in LES mode (CD-adapco, 2014). $\Psi$ is defined as

$$\Psi^2 = \min \left\{ 100, \frac{1 - C_{b1} \left( (f_{t2} + (1 - f_{t2}) f_{\nu2}) / 0.424 \kappa^2 C_{w1} \right)}{(1 - f_{t2}) f_{\nu1} f_{\nu3}} \right\}$$

(A.25)

Finally, the grid filter width $\Delta$ is given as

$$\Delta = f_{\nu} C_{s} V^{1/3}$$

(A.26)

$$f_{\nu} = \begin{cases} 1 & \text{no damping} \\ 1 - \exp \left( - \frac{y^+}{25} \right) & \text{standard} \end{cases}$$

(A.27)

where $C_s=0.1$, and $y^+ = \frac{d\sigma_{xx}}{\mu}$, where the friction velocity is $u_\tau = \sqrt{\frac{\tau_w}{\rho}}$, the wall shear stress is $\tau_w = \mu \frac{du}{dy}$, $u$ is the velocity component tangent to the wall located at the cell center, and $y$ is the direction normal to the wall.
Hydropower is the harnessing of energy from Earth’s solar-driven hydrological cycle for the generation of power. Historically, hydropower has been used by civilizations for thousands of years. From the use of water wheels for the direct operation of grinding stones in flour mills, to the irrigation of land, people have tapped into this abundant resource of energy. A revolution in technology came about in the late 19th century when an electric generator was driven by a hydraulic turbine, providing the first source of hydroelectric power.

Hydropower is the most proven renewable energy technology, supplying the world with 16% of its electricity. Although a mature technology, hydroelectric generation still shows great promise for expansion and improvement. Modern computational technology and fluid dynamics expertise has led to substantial improvements in modern turbine design and performance. Cavitation, the formation and subsequent collapse of vapor in liquid, which is initiated when the liquid drops below a critical pressure, has always presented a problem in hydroturbines. It causes performance degradation, erosion, damage, vibration, and noise. Multiphase Computational Fluid Dynamics (CFD) has been utilized in recent years to simulate cavitating flow in many industrial problems. CFD of cavitating flow in hydroturbines is still in its infancy, although it has the potential to provide unique benefits to the hydropower industry.

One such benefit is the ability to compute and analyze the steady and unsteady cavitating flow dynamics in the full-scale turbine just as easily as in the small-scale model tests. Experiments are usually conducted on small-scale models, which do not experience the same gravitational effects as the full-scale machine. The results are then estimated at full-scale, although due to the inability to scale hydrostatic pressure effects on cavitation, current estimates do not account for cavitation in
the turbine. Thus, at model test operating points for which large amounts of vapor form, estimates of the full-scale machine are insufficient.

Steady and unsteady multiphase CFD results are presented for a range of cavitating flows in both model and full-scale hydroturbines. The vapor content, performance, and flow variable fluctuations are shown to vary between the two scales. This variation becomes more significant as cavitation occurs to a greater extent. For some conditions, a diffuser downstream of the turbine displays significant cavitation in the model. This cavitation in the model does not appear in the full-scale machine, and it leads to considerably more losses in the model.

Other interesting cavitation features are also numerically resolved at both scales. A pressure imbalance results in asymmetric vapor distribution about the turbine, leading to more extensive growth and collapse of vapor. At certain operating conditions, unique pressure oscillations appear in the machine, likely caused by vapor downstream of the turbine. Additionally, vapor content on the turbine blades is affected by their interaction with stationary vanes located upstream. Finally, large maximum pressure spikes, due to cavity collapse, are observed on the turbine blades, and are shown to be a potential source of cavitation damage and erosion.

Multiphase CFD is shown to be an accurate and effective technique for simulating and analyzing cavitating flow in hydroturbines. It is recommended that it be used as an industrial tool to supplement model cavitation experiments, and as a research tool to investigate mechanisms of cavitating hydraulic turbines that are not understood.
Bibliography


Vita
Daniel J. Leonard

Daniel Leonard was born in Philadelphia, Pennsylvania, to James and Rosalie Leonard. Daniel attended Central High School in Philadelphia, where he obtained a diverse education and played soccer. After graduating from Central, he attended The Pennsylvania State University, where he developed an interest in fluid mechanics, and graduated in 2007 with a Bachelor of Science degree in Aerospace Engineering.

Daniel then continued his education at Penn State, pursuing a Master of Science in Aerospace Engineering. He conducted experimental research on the aeroacoustics of unvoiced speech sound production, under Dr. Michael Krane in the Applied Research Lab, and completed his thesis and degree in 2010. Following graduation, Daniel decided to continue towards his doctorate at Penn State and the Applied Research Lab. At this time he began research on computing cavitating flow in hydroturbines under Dr. Jules Lindau. He was also awarded an external fellowship from the Hydro Research Foundation, which helped support his doctorate.

His current research interests include fluid mechanics, acoustics, aeroacoustics, cavitation, and renewable energy. During his career, he hopes to develop and use both computational and experimental methods to conduct research in fluid mechanics across a wide range of problems and scales.