Investigating Inducer Performance over a Wide Range of Operating Conditions

David Tate Fanning
Brigham Young University

Follow this and additional works at: https://scholarsarchive.byu.edu/etd

Part of the Engineering Commons

BYU ScholarsArchive Citation
Fanning, David Tate, "Investigating Inducer Performance over a Wide Range of Operating Conditions" (2019). Theses and Dissertations. 7732.
https://scholarsarchive.byu.edu/etd/7732

This Dissertation is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact ellen_amatangelo@byu.edu.
Investigating Inducer Performance over a Wide Range of Operating Conditions

David Tate Fanning

A dissertation submitted to the faculty of
Brigham Young University
in partial fulfillment of the requirements for the degree of
Doctor of Philosophy

Steven E. Gorrell, Chair
R. Daniel Maynes
Dale R. Tree
Julie Crockett

Department of Mechanical Engineering
Brigham Young University

Copyright © 2019 David Tate Fanning
All Rights Reserved
ABSTRACT

Investigating Inducer Performance over a Wide Range of Operating Conditions

David Tate Fanning
Department of Mechanical Engineering, BYU
Doctor of Philosophy

Inducer performance is investigated for a variety of inducer geometries operating at multiple flow conditions using computational fluid dynamics. Inducers are used as a first stage in turbopumps to minimize cavitation and allow the pump to operate at lower inlet head conditions. The formation of inlet flow recirculation or backflow in the inducer occurs at low flow conditions and can lead to instabilities and cavitation-induced head breakdown. Backflow formation is often attributed to tip leakage flow. The performance of an inducer with and without tip clearance is examined. Removing the tip clearance eliminates tip leakage flow; however, backflow is still observed. Analysis suggests that blade inlet diffusion, not tip leakage flow, is the fundamental mechanism leading to the formation of backflow.

Performance improvements in turbopump systems pumping cold water have been obtained through implementation of a recirculation channel called a stability control device (SCD). However, many inducers actually pump cryogenic fluids, such as liquid hydrogen. To determine the real-world effects of SCD implementation, inducer performance at on and off design flow coefficients with and without an SCD were modeled with liquid hydrogen as the working fluid. Relevant thermodynamic effects present in liquid hydrogen at cryogenic temperatures are considered. The results reveal that the SCD yields marginal changes in the head coefficient. However, a stabilizing effect occurs at all considered flow coefficients, where a reduction in backflow occurs over much of the pump operational range. This occurs due to the SCD maintaining consistent, low incidence angles at the inducer leading edge.

The final consideration of this work is the acceleration of an inducer from rest to the operating rotational rate. Rapid acceleration of rocket engine turbopumps during start-up imparts significant transient effects to the resulting flow field, causing pump performance to vary widely when compared to quasi-steady operation. A method to simulate turbopump start-up using CFD is developed and presented. The defined outlet pressure is modified based on the difference between simulation inlet pressure and target inlet pressure of a previous simulation. This process is repeated until simulation inlet pressure is essentially constant during start-up. Using this novel simulation method, the performance of a centrifugal turbopump during start-up is simulated. Analysis suggests this simulation method provides a reasonable prediction of cavitation formation and inducer performance.

Keywords: cavitation, backflow, start-up, inducer, CFD, suction performance, cryogenic, thermal suppression
I would like to express sincere thanks to Dr. Steven Gorrell, Dr. Daniel Maynes, and Kerry Oliphant. Their guidance and expertise was invaluable, and this work would not have been possible without them. I would also like to thank Jamin Bitter for his advice and assistance in generating meshes and analyzing simulation data. I am grateful to Concepts NREC for providing the funding for this research project and allowing publication of the data. I am also very grateful for the financial support provided by the Utah NASA Space Grant Consortium. This work would not have been possible without the Star-CCM+ licenses provided by CD-Adapco and the CRUNCH license provided by CRAFT Tech nor the computing resources and support provided by the BYU Office of Research Computing. Finally, I would like to thank my wife for her support during my studies.
# TABLE OF CONTENTS

**LIST OF TABLES** ................................................................. vii

**LIST OF FIGURES** ............................................................ viii

**NOMENCLATURE** .............................................................. xv

Nomenclature ........................................................................ xv

**Chapter 1** Introduction ......................................................... 1

1.1 Pump Background ............................................................ 2
1.2 Cavitation Instabilities ..................................................... 6
1.3 Secondary Flows ............................................................. 12
1.4 Thermal Effects ............................................................... 16
1.5 Stability Control Device ................................................... 18
1.6 Contributions from the Present Work .................................. 21
1.7 Dissertation Organization ............................................... 21

**Chapter 2** Methodology ......................................................... 23

2.1 Star-CCM+ ........................................................................ 23
2.1.1 Meshing ....................................................................... 23
2.1.2 Time Discretization ....................................................... 27
2.1.3 Time and Grid Independence ......................................... 27
2.1.4 Rotation ...................................................................... 28
2.1.5 Cavitation Modeling ..................................................... 29
2.1.6 Turbulence Modeling .................................................... 30
2.1.7 Boundary Conditions ................................................... 30
2.1.8 Validation .................................................................... 31
2.2 CRUNCH CFD ................................................................... 34
2.2.1 Meshing ...................................................................... 34
2.2.2 Time Discretization ....................................................... 36
2.2.3 Grid Independence and Validation ................................. 38
2.2.4 Rotation ..................................................................... 40
2.2.5 Cavitation Modeling ..................................................... 41
2.2.6 Turbulence Modeling .................................................... 44
2.2.7 Boundary Conditions ................................................... 45
2.2.8 Running ...................................................................... 46

**Chapter 3** Contributions of Tip Leakage and Inlet Diffusion on Inducer Backflow ........................................... 48

3.1 Contributing Authors and Affiliations ................................. 48
3.2 Abstract .......................................................................... 48
3.3 Introduction ..................................................................... 49
3.4 Methods .......................................................................... 52
| Chapter 4  | Cryogenic Cavitation Performance of an Axial Inducer with a Stability Control Device | 76 |
| 4.1       | Contributing Authors and Affiliations                                         | 76 |
| 4.2       | Abstract                                                                    | 76 |
| 4.3       | Introduction                                                                 | 77 |
| 4.4       | Methods                                                                     | 80 |
| 4.5       | Results                                                                      | 84 |
| 4.5.1     | Simulation Validation                                                        | 84 |
| 4.5.2     | Suction Performance                                                          | 85 |
| 4.5.3     | Rotordynamic Forces                                                          | 88 |
| 4.5.4     | Stability                                                                    | 90 |
| 4.5.5     | Backflow Formation                                                           | 97 |
| 4.6       | Conclusions                                                                  | 100 |

| Chapter 5  | Cavitation Inception and Performance of a Centrifugal Impeller During Start-Up | 103 |
| 5.1       | Contributing Authors and Affiliations                                         | 103 |
| 5.2       | Abstract                                                                    | 103 |
| 5.3       | Introduction                                                                 | 104 |
| 5.4       | CFD Modeling                                                                 | 106 |
| 5.4.1     | Spatial Discretization                                                       | 106 |
| 5.4.2     | Cavitation Model                                                             | 108 |
| 5.4.3     | Turbulence Model                                                             | 109 |
| 5.4.4     | Boundary and Initial Condition Specification                                | 110 |
| 5.4.5     | Processing                                                                   | 112 |
| 5.5       | Results                                                                      | 114 |
| 5.6       | Conclusions                                                                  | 120 |

| Chapter 6  | Conclusions                                                                  | 123 |
| 6.1       | Recommendations for Further Investigation                                    | 126 |

REFERENCES                                                                   | 127 |

Appendix A  | CRUNCH Simulation Tutorial                                                    | 134 |
<p>| A.1       | Simulation Steps                                                             | 134 |
| A.2       | Mesh Generation                                                              | 134 |
| A.3       | Mesh Preprocessing                                                           | 135 |
| A.4       | Single Phase Module Input File Setup                                         | 136 |
| A.5       | Single Phase Module Input File                                               | 138 |
| A.6       | Single Phase Global Input File Setup                                         | 158 |
| A.7       | Single Phase Global Input File                                               | 158 |
| A.8       | Run Single Phase Simulation                                                  | 159 |</p>
<table>
<thead>
<tr>
<th>A.9</th>
<th>Supercomputer Run File</th>
<th>160</th>
</tr>
</thead>
<tbody>
<tr>
<td>A.10</td>
<td>Post Process Single Phase Simulation</td>
<td>163</td>
</tr>
<tr>
<td>A.11</td>
<td>Cavitating Module Input File Setup</td>
<td>163</td>
</tr>
<tr>
<td>A.12</td>
<td>Cavitating Module Input File</td>
<td>164</td>
</tr>
<tr>
<td>A.13</td>
<td>Cavitating Global Input File Setup</td>
<td>184</td>
</tr>
<tr>
<td>A.14</td>
<td>Run Cavitating Simulation</td>
<td>184</td>
</tr>
<tr>
<td>A.15</td>
<td>Post Process Cavitating Simulation</td>
<td>184</td>
</tr>
<tr>
<td>A.16</td>
<td>Repeat Cavitating Simulation Process</td>
<td>185</td>
</tr>
<tr>
<td>A.17</td>
<td>Simulation Errors</td>
<td>186</td>
</tr>
<tr>
<td>A.18</td>
<td>POSTCRUNCH Output Error</td>
<td>187</td>
</tr>
</tbody>
</table>
2.1 Inducer performance for converged solutions with varying time steps for the 7° inducer considered in Chapter 3, at $\phi = 0.07$ and $\sigma \approx 0.022$. Percent changes are calculated from the values corresponding to the time step of $1.4 \times 10^{-5}$ s. From [8].

3.1 Grid convergence index comparison for varying mesh sizes. These values loosely represent error bands for data extracted from the numerical simulations.

4.1 Geometric details of considered flat plate inducer.

4.2 Grid convergence index comparison for varying mesh sizes at two operational points. These values loosely represent error bands for data extracted from the numerical simulations.

4.3 Local incidence averaged along span and all NPSH values for all simulated flow coefficients.

5.1 Grid convergence index comparison for three mesh sizes. These values loosely represent error bands for data extracted from the numerical simulations.
LIST OF FIGURES

1.1 (a) Image of a standard rocket turbopump inducer with integral centrifugal downstream impeller. (b) Cross-sectional diagram of a standard rocket turbopump inducer with integral centrifugal downstream impeller. From [2]. ......................................................... 2
1.2 Image of standard rocket turbopump inducer operating under cavitating conditions . . . . . 3
1.3 Typical head-flow curves comparing the performance of two pumps over a range of operating conditions ................................................................. 4
1.4 Typical cavitating performance curves using different suction performance measures for a constant flow coefficient: (a) using cavitation number and (b) using suction specific speed. From [8]. ........................................................................................................ 7
1.5 Cavitation inception in an inducer; (a) an isosurface corresponding to a vapor fraction of 10% shows cavity formation begins at the leading edge near the blade tips, and (b) streamlines illustrating the relationship between the tip vortex and cavitation formation. 8
1.6 Isosurface views of of $\phi_{vapor} = 0.1$ at varying $\sigma$. Cavity volumes increase with decreasing inlet pressure and reduce inducer performance: (a) $\sigma = 0.4903$; (b) $\sigma = 0.2085$; (c) $\sigma = 0.0665$; (d) $\sigma = 0.0179$. ................................................................. 9
1.7 Sequence of contour plots of volume fraction of vapor at 98% span for a three blade inducer operating at $\sigma \approx 0.019$ over a period of 1.7 revolutions. From [2]. ......................... 10
1.8 Schematic of cavitation-caused flow blockage. From [6]. .............................................. 12
1.9 Typical backflow upstream propagation: (a) at high flow coefficient and (b) at low flow coefficient. Blue regions identify flow with a velocity component opposite that of the core flow. ................................................................. 13
1.10 Contour plot of volume fraction of vapor in an inducer experiencing upstream cavitation. This cavity formation can occur due to backflow restricting the upstream flow area and accelerating the flow. From [4]. ......................................................... 14
1.11 Lateral view of impeller illustrating tip leakage flow and the formation of backflow. Adapted from [6]. ................................................................. 15
1.12 Vapor pressure of liquid hydrogen and liquid oxygen at various temperatures. Vapor pressure is highly dependent on fluid temperature for these liquids. ......................... 16
1.13 Cross section of an inducer equipped with an SCD. High energy fluid is captured behind the leading edge of the inducer, is diffused, and reintroduced into the upstream flow. ................................................................. 19
2.1 Representative images of tetrahedral and polyhedral cells .............................................. 24
2.2 (a) Cross-section of typical structured mesh with hexahedral cells. (b) Cross-section of typical unstructured mesh with polyhedral cells. ......................................................... 25
2.3 Detail image of cross-section of typical unstructured mesh to highlight prism layer cells . . . 26
2.4 Star-CCM+ computational domain of an axial inducer with labeled boundary conditions used in Chapter 3 ................................................................. 31
2.5 Star-CCM+ computational domain of an axial inducer with labeled boundary conditions used in Chapter 5 ................................................................. 32
2.6 Comparison of simulated cavitating performance to experimental data. From [8]. (a) $7^\circ$ inducer with no SCD. (b) $7^\circ$ inducer with an SCD. (c) $9^\circ$ inducer with an SCD. . . . 33
2.7 (a) Cross-section of CRUNCH mesh with no SCD. (b) Cross-section of CRUNCH mesh with SCD. ................................................................. 35
2.8 (a) Tip detail of CRUNCH mesh with no SCD. (b) Tip detail of CRUNCH mesh with SCD. ................................................................. 36
2.9 (a) Detail of upstream SCD interface. (b) Detail of downstream SCD interface. ........ 37
2.10 (a) Tip detail of 16 million cell mesh with no SCD. (b) Tip detail of 1 million cell mesh with no SCD. ................................................................. 39
2.11 Cavitating performance curves comparing experimental data to simulations generated with multiple mesh sizes .................................................. 40
2.12 Cavitating performance curve comparing experimental data to simulation data using the 9 million cell mesh. All simulation data was shifted by a NPSH reduction of 50 ft. 41
2.13 Cavitating performance curves comparing experimental data to simulation data while varying cavitation time constants ....................................... 43
2.14 Image of CRUNCH inducer shroud boundary surfaces showing slip and no slip surfaces. The rotor blades, rotor hub, and rotor shroud are no slip walls. The upstream rotor shroud and upstream rotor hub are slip walls. ................................................................. 46

3.1 Lateral view of impeller illustrating tip leakage flow and the formation of backflow. Adapted from [6]. ................................................................. 50
3.2 Mid-plane section of the tip clearance inducer geometry mesh ................................................................. 53
3.3 Tip mesh detail of both considered inducer geometries: (a) TC mesh and (b) NTC mesh 54
3.4 Non-dimensional backflow penetration distance as a function of flow coefficient. The solid line is a fit to the NTC data while the dashed line is a fit to the TC data. At all \( \phi < 0.7 \) the NTC inducer experiences greater backflow upstream penetration. At \( \phi > 0.7 \) the TC inducer exhibits slightly greater backflow upstream penetration, although the backflow is vanishing for both inducers. ................................................................. 57
3.5 Non dimensional backflow mass flow at the leading edge as a function of flow coefficient. The TC inducer exhibits greater backflow mass flows at \( \phi > 0.042 \). ....... 58
3.6 Nondimensional axial velocity profiles as a function of percent span (hub to shroud) for axial upstream locations: (a) \( z/D_{tip} = 0.5 \), (b) \( z/D_{tip} = 1.5 \), and (c) \( z/D_{tip} = 3 \) .... 59
3.7 Contour plots of axial velocity magnitude in a cross sectional view of both inducer geometries operating at \( \phi = 0.042 \). Axial velocity is colored with a red-blue color bar. Backflow extends further upstream in the NTC case: (a) NTC inducer and (b) TC inducer. ................................................................. 60
3.8 Streamlines plotted just upstream of the leading edge of the TC inducer operating at \( \phi = 0.042 \). \( \lambda_2 \) criterion is colored with a red-blue color bar. Regions where \( \lambda_2 < 0 \) can be interpreted as vortex regions. Regions where \( \lambda_2 \geq 0 \) have no physical interpretation. This plot differentiates the tip vortex from the backflow. The NTC inducer has been truncated at \( r/R = 0.995 \) to allow for better visualization of the tip vortex: (a) NTC inducer and (b) TC inducer. ................................................................. 62
3.9 Normalized tip vortex volume as a function of flow coefficient for both inducers. At all flow coefficients, the NTC tip vortex is larger than the TC tip vortex. .... 63
3.10 Normalized tip vortex circulation as a function of flow coefficient for both inducers explored. At all flow coefficients, the TC tip vortex has greater circulation than the NTC tip vortex. ................................................................. 64
3.11 Pressure recovery plotted as a function of flow coefficient. The NTC inducer experiences a greater pressure recovery, at all considered flow coefficients.

3.12 Non-dimensional performance curves for both inducer geometries. The solid line is a fit to the NTC data while the dashed line is a fit to the TC data. At all $\phi$ the NTC inducer experiences greater head production than the TC inducer.

3.13 Typical backflow formation in a centrifugal impeller. The pressure distribution of the impeller has forced the separated flow upstream of the leading edge of the blades. From [86].

3.14 Non-dimensional backflow penetration as a function of $AR$ for all simulated flow coefficients. Polynomial curve fits for each data set are shown. The curve fit for the NTC data has a $R^2 = 0.9998$, while the TC data fit has an $R^2 = 0.9999$.

3.15 Non-dimensional backflow mass flow as a function of $AR$ for all simulated flow coefficients. Linear curve fits for each data set are shown. The curve fit for the NTC data has a $R^2 = 0.9982$, while the TC data fit has an $R^2 = 0.9981$.

3.16 $AR_c$ as a function of cavitation number. A solid line is fit through all the data, for which $R^2 = 0.9431$.

3.17 Isosurface views of of $\phi_{vapor} = 0.1$ for both inducer geometries. Cavity volumes affect upstream flow conditions and introduce error into $AR_c$ calculations: (a) NTC inducer, $AR = 2.9, AR_c = 2.9, \sigma = 0.025$; (b) NTC inducer, $AR = 2.9, AR_c = 2.9, \sigma = 0.157$; (c) TC inducer, $AR = 2.9, AR_c = 2.9, \sigma = 0.062$; (d) TC inducer, $AR = 2.9, AR_c = 2.9, \sigma = 0.222$.

4.1 Schematic illustration of a flat plate inducer with an SCD.

4.2 Considered inducer geometries and corresponding meshes.

4.3 Non cavitating performance curve of the considered inducer over the range of considered flow coefficients. Experimental results from Meng [29] are compared to CRUNCH CFD results for both SCD and no SCD configurations.

4.4 Cavitation breakdown curves for all considered flow coefficients. Experimental results from Meng [29] are compared to CRUNCH CFD results for both SCD and no SCD configurations at $\phi = 0.117$ and $\phi = 0.107$. Meng did not test the inducer at 21.1 K for $\phi = 0.097$ or $\phi = 0.07$.

4.5 Lateral force coefficient acting on both inducer geometries for all considered flow coefficients. SCD implementation reduces lateral forces on the inducer for all simulated flow coefficients.

4.6 Incidence along blade span at $\phi = 0.117$ for the highest and lowest simulated values of NPSH.

4.7 Incidence along blade span at $\phi = 0.107$ for the highest and lowest simulated values of NPSH.

4.8 Incidence along blade span at $\phi = 0.097$ for the highest and lowest simulated values of NPSH.

4.9 Incidence along blade span at $\phi = 0.08$ for the highest and lowest simulated values of NPSH.
4.10 Incidence along blade span at $\phi = 0.07$ for the highest and lowest simulated values of NPSH. ......................................................... 95
4.11 SCD mass flow gain factor as a function of $AR_c$ for all simulated flow coefficients. A curve fit of the form $K_{SCD} = 0.3737 \times AR_c + 0.5789$ is also shown. ......................... 98
4.12 Upstream backflow penetration plotted as a function of $AR_c$, which considers the reduction in flow area due to cavitation blockage. The SCD significantly limits backflow penetration at all flow coefficients below $\phi = 0.117$. ............................................ 99
4.13 $AR_c$ plotted as a function of cavitation number. A solid line is fit through the data for each flow coefficient. .............................................................. 100

5.1 Mid-plane section of the mesh .................................................. 107
5.2 Tip detail of a mid-plane section of the mesh ................................. 107
5.3 Impeller rotation rate during start-up for the considered pump ...................... 111
5.4 Flow chart describing simulation iteration process ............................... 113
5.5 Normalized difference between actual inlet total pressure and target inlet total pressure during start-up for each simulation iteration ............................... 115
5.6 Normalized outlet static pressure during start-up for each simulation iteration ........ 115
5.7 Head coefficient produced by the impeller during start-up for each simulation iteration 117
5.8 Isosurface of 30% vapor fraction showing cavitation formation on the pressure side of the impeller blades in Iteration 5. ...................................... 118
5.9 Contour plot of vapor fraction on a cross section of the domain. This provides another view of the cavitation formation on the pressure side of the impeller blades in Iteration 5. .......................... 119
5.10 Normalized cavity volume during start-up for each simulation iteration .............. 119
5.11 Resulting flow coefficient for the impeller during start-up for each simulation iteration 120
5.12 Normalized RMS force on the impeller during start-up for each simulation iteration .... 121
# NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AR</td>
<td>Area ratio, $\sin \beta_b / \sin \beta$</td>
</tr>
<tr>
<td>AR&lt;sub&gt;c&lt;/sub&gt;</td>
<td>Cavitating area ratio, $1 - T_c$</td>
</tr>
<tr>
<td>AR&lt;sub&gt;crit&lt;/sub&gt;</td>
<td>Critical area ratio</td>
</tr>
<tr>
<td>A&lt;sub&gt;flow&lt;/sub&gt;</td>
<td>Throat area of blade passage less cavitation blockage</td>
</tr>
<tr>
<td>A&lt;sub&gt;inlet&lt;/sub&gt;</td>
<td>Inlet area of a single blade passage</td>
</tr>
<tr>
<td>A&lt;sub&gt;LE&lt;/sub&gt;</td>
<td>Frontal area of inducer at leading edge</td>
</tr>
<tr>
<td>A&lt;sub&gt;throat&lt;/sub&gt;</td>
<td>Throat area of a blade passage</td>
</tr>
<tr>
<td>C&lt;sub&gt;axial&lt;/sub&gt;</td>
<td>Axial velocity</td>
</tr>
<tr>
<td>C&lt;sub&gt;p,max&lt;/sub&gt;</td>
<td>Maximum pressure recovery</td>
</tr>
<tr>
<td>D&lt;sub&gt;tip&lt;/sub&gt;</td>
<td>Inducer Blade tip diameter</td>
</tr>
<tr>
<td>F&lt;sub&gt;RMS&lt;/sub&gt;</td>
<td>RMS force on an inducer</td>
</tr>
<tr>
<td>g</td>
<td>Acceleration due to gravity</td>
</tr>
<tr>
<td>i</td>
<td>Incidence angle, $i = \beta_b - \beta$</td>
</tr>
<tr>
<td>L&lt;sub&gt;p&lt;/sub&gt;</td>
<td>Backflow upstream penetration measured from inducer leading edge</td>
</tr>
<tr>
<td>˙m&lt;sub&gt;back&lt;/sub&gt;</td>
<td>Mass flow in fluid backflow region</td>
</tr>
<tr>
<td>˙m&lt;sub&gt;in&lt;/sub&gt;</td>
<td>Inlet mass flow</td>
</tr>
<tr>
<td>P</td>
<td>Fluid pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;shroud&lt;/sub&gt;</td>
<td>Maximum static pressure on shroud wall</td>
</tr>
<tr>
<td>NPSH</td>
<td>Net Positive Suction Head</td>
</tr>
<tr>
<td>P&lt;sub&gt;00&lt;/sub&gt;</td>
<td>Inlet total pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;00, target&lt;/sub&gt;</td>
<td>Target inlet total pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;01&lt;/sub&gt;</td>
<td>Outlet total pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;inlet&lt;/sub&gt;</td>
<td>Inlet static pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;out&lt;/sub&gt;</td>
<td>Outlet static pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;s&lt;/sub&gt;</td>
<td>Outlet static pressure defined by radial equilibrium boundary condition</td>
</tr>
<tr>
<td>P&lt;sub&gt;s, hub, def&lt;/sub&gt;</td>
<td>Defined static pressure</td>
</tr>
<tr>
<td>P&lt;sub&gt;v&lt;/sub&gt;</td>
<td>Fluid vapor pressure</td>
</tr>
<tr>
<td>Q</td>
<td>Inlet volumetric flow rate</td>
</tr>
<tr>
<td>r</td>
<td>Radial position</td>
</tr>
<tr>
<td>r&lt;sub&gt;i&lt;/sub&gt;</td>
<td>Minimum radial position relative to axis of rotation</td>
</tr>
<tr>
<td>R</td>
<td>Shroud radius</td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
</tr>
<tr>
<td>U&lt;sub&gt;tip&lt;/sub&gt;</td>
<td>Impeller blade tip speed</td>
</tr>
<tr>
<td>V</td>
<td>Velocity</td>
</tr>
<tr>
<td>V&lt;sub&gt;θ&lt;/sub&gt;</td>
<td>Tangential velocity</td>
</tr>
<tr>
<td>V&lt;sub&gt;vapor&lt;/sub&gt;</td>
<td>Total vapor volume</td>
</tr>
<tr>
<td>V&lt;sub&gt;tip vortex&lt;/sub&gt;</td>
<td>Tip vortex volume</td>
</tr>
<tr>
<td>z</td>
<td>Upstream axial position measured from inducer leading edge</td>
</tr>
<tr>
<td>β</td>
<td>Inducer inlet flow angle from the tangential</td>
</tr>
<tr>
<td>β&lt;sub&gt;b&lt;/sub&gt;</td>
<td>Inducer inlet blade angle from the tangential</td>
</tr>
<tr>
<td>δ&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Cavitation blockage displacement thickness</td>
</tr>
<tr>
<td>δ&lt;sub&gt;min&lt;/sub&gt;</td>
<td>Minimum cavitation blockage displacement thickness</td>
</tr>
<tr>
<td>φ</td>
<td>Flow coefficient, $\phi = ˙m_{in} / (\rho A_{LE} U_{tip})$</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>----------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>$\phi_{vapor}$</td>
<td>Volume fraction of vapor</td>
</tr>
<tr>
<td>$\psi$</td>
<td>Head coefficient, $\psi = \frac{P_{b1} - P_{b0}}{\rho U_{tip}^2}$</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Tip clearance ratio</td>
</tr>
<tr>
<td>$T_c$</td>
<td>Throat Cavitation Blockage, $T_c = \int \phi_{vapor} dA / A_{throat}$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Cavitation number, $\sigma = \frac{P_{b0} - P_v}{\frac{1}{2} \rho U_{tip}^2}$</td>
</tr>
<tr>
<td>$\omega_\theta$</td>
<td>Averaged tangential vorticity</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Inducer angular velocity</td>
</tr>
</tbody>
</table>
CHAPTER 1. INTRODUCTION

An integral part in high suction performance turbopumps is the inducer. An inducer is an axial rotor that revolves at high angular velocities, imparting higher pressures and velocities to the working fluid. As seen in Figure 1.1, an inducer is often placed immediately upstream of an impeller. The high angular velocities of an inducer can cause the local static pressure of the working fluid to drop below the vapor pressure, which can form vapor bubbles called cavitation. High suction performance refers to the ability of a pump to operate with low suction head, which corresponds to cavitating conditions. Figure 1.2 shows cavitation occurring upstream of the inducer and following the tips of the inducer blades. Cavitation formation is a major flow event that can lead to unstable operating conditions, reduced performance, or even total pump failure. The purpose of an inducer placed upstream of an impeller is to pressurize the fluid sufficiently to avoid cavitation formation in the rest of the pump system, and avoid its adverse effects [1]. Despite inclusion of an inducer and specific design considerations to operate under cavitating conditions, cavitation volumes can still form which limits inducer performance and reduces the pump operating range.

Designing an inducer to operate stably while minimizing cavity formation is a complex task. Minute changes in many design parameters including inlet blade angle, tip clearance, blade number, turning angle, sweep, inlet eye diameter, design point incidence, and leading edge shape can have significant effects on performance [3–5]. This work focuses on how three parameters affect inducer performance and cavitation formation, namely tip clearance, stability control device implementation with liquid hydrogen, and transient start-up.

The remainder of Chapter 1 provides background information to provide a basis of understanding for Chapters 2 - 5. More specifically, overviews of relevant pump parameters, cavitation instabilities, thermal effects, stability control device development and performance are provided along with contributions and organization of this dissertation.
1.1 Pump Background

A review of relevant pump parameters is necessary before the analysis of inducer performance is presented. Because pump geometry, flow rates, operating pressures, and rotational rates vary widely, pump performance is generally described with dimensionless numbers to allow for easy comparison of a wide range of pumps. Flow coefficient (\(\phi\)), shown in Eq. 1.1 describes the mass flow through the pump, where \(m_{in}\) is the mass flow rate at pump inlet, \(\rho\) is the density of the working fluid, \(A_{LE}\) is the cross-sectional area of the pump at the leading edge of the blades, and \(U_{tip}\) is the blade tip speed. Flow coefficient can also be expressed as the ratio of the mean inlet meridional velocity to the blade tip speed (\(\phi = C_m/U_{tip}\)).
The main measure of pump performance is the head coefficient ($\psi$), which is a measure of the pressure rise produced by the pump normalized by the fluid kinetic energy at the blade tip. Head coefficient is defined in Eq. 1.2, where $P_{01}$ is the total pressure at the outlet of the pump, and $P_{00}$ is the total pressure at the pump inlet.

$$\psi = \frac{P_{01} - P_{00}}{\rho U_{tip}^2}$$  \hspace{1cm} (1.2)$$

Non-cavitating pump performance is often presented as a performance, or head-flow curve, which plots flow coefficient against non-cavitating head coefficient. Non-cavitating head coefficient describes the pump head coefficient when fluid pressures are sufficiently high such that cavitation does not form in the pump. Figure 1.3 shows examples of two performance curves. The inducer described by the solid line generates a higher head coefficient at all flow coefficients. Typically, head increases with decreasing flow coefficient. As flow coefficient decreases, the fluid load increases on the blade, allowing the inducer to impart more work to the fluid, and increase the head produced by the pump. However, as flow coefficient continues to decrease, the pump
Figure 1.3: Typical head-flow curves comparing the performance of two pumps over a range of operating conditions

Blades can stall, resulting in significantly reduced head coefficient. This is evident in Fig. 1.3 for the inducer described by the solid line, as head begins to flatten at the lowest two flow coefficients. If this inducer were allowed to operate at even lower flow coefficients than shown in Fig. 1.3, an inflection point would likely occur, and the performance curve would exhibit a positive slope with further decreases in flow coefficient. Positive slopes in the performance curve are typically characterized by pump surging, with significant flow rate and pressure oscillations.

Blade loading, or the amount of work each blade does on the fluid, is defined as the difference in static pressure between blade suction and pressure surfaces. Blade loading is related to the incidence angle \(i\). Incidence is defined in Eq. 1.3, and is the difference between the blade angle, \(\beta_b\), and the flow angle, \(\beta\). For axial inducers, the flow angle is often approximated as \(\beta = \arctan(\phi)\) [6], as there is no tangential velocity upstream of the inducer. The incidence angle is critical in the design of high suction performance pumps, as it affects how the fluid loads the blade.

\[
i = \beta_b - \beta \tag{1.3}
\]
Large incidence angles can induce detrimental secondary flows such as backflow, which can cause unstable pumping conditions even in the absence of cavitation [6–8]. Incidence angles near zero for a symmetric airfoil correspond to an unloaded blade, which can promote flow separation and cavitation formation on pressure and suction sides of the blades while reducing pump performance by limiting the amount of work transferred to the fluid. Typical design incidence angles for high suction performance turbopumps are in the range of $2^\circ – 4^\circ$ [3].

Turbopump cavitating performance can be described by the amount of suction head required to operate the pump. Due to the low pressure at the pump inlet caused by the high angular velocities of turbopumps, it is desirable that these pumps operate with as low a suction head as possible. If one pump has a greater suction performance than another, it means that the higher performance pump is able to operate at a lower suction head. Suction performance is dependent on minimizing bulk (average) velocity upstream of an inducer [4]. The net positive suction head (NPSH) is defined in Eq. 1.4, and is one measure to describe the inlet suction head of the pump where $P_v$ is the fluid total pressure, and $g$ is acceleration due to gravity. The cavitation number, $\sigma$, defined in Eq. 1.5 is essentially equivalent to NPSH, but uses tip speed instead of gravity to normalize the inlet pressure head. Cavitation number is inversely related to the cavity volume, meaning a high cavitation number describes low cavity volumes, or a low propensity to cavitate. A third common measure of suction performance is the suction specific speed ($N_{ss}$) defined in Eq. 1.6, where $\omega$ is the inducer angular velocity. The suction specific speed is inversely related to NPSH and the cavitation number, and can also be easily defined using the NPSH, as shown in Eq. 1.7, where $Q$ is the inlet volumetric flow rate. A high performance inducer will have a large $N_{ss}$, meaning suction performance scales with $N_{ss}$. The general trend is that $N_{ss}$ rises dramatically with reduction in flow coefficient [4]. Therefore, significantly improved pump performance is possible at lower flow coefficients. Unfortunately, operating at low flow coefficients is difficult due to cavitation, so inducers are designed to operate at a specific design flow coefficient, typically above 0.06. These design flow coefficients provide the best suction performance while maintaining a sufficient margin of safety from the cavitation-induced instabilities.

\[
NPSH = \frac{P_{bo} - P_v}{\rho g}
\]  

(1.4)
\[ \sigma = \frac{P_{00} - P_v}{\frac{1}{2} \rho U_{tip}^2} \]  

(1.5)

\[ N_{ss} = \frac{\omega \sqrt{\frac{m_{in}}{\rho}}}{((P_{00} - P_v)/\rho)^{0.75}} \]  

(1.6)

\[ N_{ss} = \frac{\omega \ast Q^{0.5}}{NPSH^{0.75}} \]  

(1.7)

Cavitating performance is presented differently than non-cavitating performance. Cavitating performance curves plot the head coefficient against either NPSH, cavitation number, or \( N_{ss} \) at a constant flow coefficient. Figure 1.4 provides typical cavitating performance curves using the cavitation number and \( N_{ss} \). A performance curve using NPSH would look exactly the same as a performance curve using the cavitation number, but the x-axis values would have different magnitudes. As can be seen in Fig. 1.4, cavitating performance typically decreases at low cavitation numbers (or NPSH), and high suction specific speeds, however, the performance degradation is not constant. For the majority of the curve, the inducer will produce a relatively constant head. However, once cavity volumes grow large enough to decrease pump efficiency, the head produced by the inducer can drop sharply with small additional decreases in inlet pressure. This sharp decline in inducer performance is known as cavitation breakdown, and is typically defined by a 3% decrease in head coefficient [3], although much larger decreases often occur.

1.2 Cavitation Instabilities

Cavitation produces adverse results in three main categories [6]. First, the material of the blade surface is damaged by the cyclic stress imparted by the collapse of vapor bubbles. Second, the performance of the pump itself can be degraded due to changes in the inlet pressure outside the design pressure range. Formation of cavities can block flow, significantly reducing pump performance, which is called cavitation breakdown. Finally, the formation of cavitation bubbles affects both the steady and unsteady response of the flow. The transient nature of cavitation can create instabilities and oscillating flow rates and pressures which can impart large shaft orbits or axial
oscillations to the inducer. The forces created by these instabilities can result in pump mechanical failure including damaged bearings and blade to casing contact.

Cavity volumes form in regions of low pressure, and typically begin formation at the inducer leading edge at the blade tip. The exact inception point often corresponds to the low pressure center of a vortex located at the blade tip that is known as the blade tip vortex [8]. Figure 1.5a depicts cavitation inception of an inducer operating at a relatively high cavitation number at the leading edge near the blade tips with an isosurface colored blue that depicts regions with a vapor volume fraction of 10%. Figure 1.5b shows the same isosurface seen in Fig. 1.5a, but plots it alongside streamlines depicting the tip vortex at the blade leading edge. The streamlines suggest that two vortices form upstream of the leading edge; one at the blade tip, and one just upstream of the inducer at a similar radial location as the blade tips. No other vortices are apparent in the
flow field from this view. The cavity volume isosurface is again shown in blue, and is located at the blade tip. The cavity volume and the core of the blade tip vortex overlap. It is clear from 1.5b that cavitation inception coincides with the tip vortex.

Figure 1.5: Cavitation inception in an inducer; (a) an isosurface corresponding to a vapor fraction of 10% shows cavity formation begins at the leading edge near the blade tips, and (b) streamlines illustrating the relationship between the tip vortex and cavitation formation.

As inlet pressure drops, cavitation number and cavity volumes grow. Cavitation growth with decreasing cavitation number can be seen in Fig. 1.6 at four different cavitation numbers: $\sigma = 0.4903, 0.2085, 0.0665,$ and $0.0179$. Cavitation volumes at $\sigma = 0.4903$ in Fig. 1.6a are initially
small and located at the blade tips. With the drop in cavitation number to $\sigma = 0.0285$ in Fig. 1.6b, cavity volumes increase, but remain near the blade tips as four separate, distinct volumes. Cavity volume increases drastically as cavitation number decreases to $\sigma = 0.0665$ in Fig. 1.6c. At this $\sigma$, the cavity volume spans from hub to tip and extends much farther along the blade and upstream of the leading edge than at higher $\sigma$, and the inducer is on the cusp of breakdown. At $\sigma = 0.0179$ in Fig. 1.6d, there are no longer four distinct cavity volumes. Cavitation completely engulfs the blades, and extends upstream of the leading edge and into the blade passages, causing complete breakdown. At this point, the slope of the cavitating performance curve is vertical. It is easy to see how increasing cavity volume can prevent an inducer from doing work on a fluid, and how cavitation can cause pump breakdown.

Figure 1.6: Isosurface views of of $\phi_{vapor} = 0.1$ at varying $\sigma$. Cavity volumes increase with decreasing inlet pressure and reduce inducer performance: (a) $\sigma = 0.4903$; (b) $\sigma = 0.2085$; (c) $\sigma = 0.0665$; (d) $\sigma = 0.0179$. 
While cavitation formation in Fig. 1.6 is circumferentially uniform and symmetric, that is not always how cavitation forms. Asymmetric cavity formation can occur and induce transient instabilities to an inducer. Two significant cavitation events that induce instabilities are cavitation surge and rotating cavitation. Cavitation surge is a system instability that involves strong flow and pressure oscillations in the direction of primary flow that correspond with in-phase cavity fluctuations on each blade, and may cause failure in the pump [1, 9, 10]. These flow and pressure oscillations are generally formed by the growth and collapse of cavitation at the inducer inlet at low, off-design flow rates. Figure 1.7 shows cavitation surge in a sequence of contour plots of vapor fraction at 98% span for a three bladed inducer simulated by Cluff [2] operating at $\sigma \approx 0.019$ over a period of 1.7 revolutions. The sharp cavity volume growth and dissipation between 1.0 and 1.7 revolutions is cavitation surge.

An estimate of the the onset condition for cavitation surge is characterized by a change in head rise and flow coefficient, and was derived with a one-dimensional analysis of an impeller controlled by a valve of resistance $R$, and a discharge tank of compliance $C$, defined in Eq. 1.8, where $\alpha$ is the speed of sound and $V$ is the tank volume. [11]. The onset condition is defined in Eq. 1.9, where $\beta^*$ is the average blade angle between the inlet and outlet of the inducer, $l$ is the chord length, $L$ is the inlet conduit length, $R$ is a measure of resistance in the control valve, and $B$ is Greitzer’s $B$ factor. Greitzer’s $B$ factor is defined in Eq. 1.10.

$$C = \frac{V}{\rho \alpha^2 A_{LE}}$$  \hspace{1cm} (1.8)
\[
\frac{\partial \psi}{\partial \phi} = \frac{1 + (1/cos\beta^*)(l/L)}{B^2 \phi R}
\]

\[
B = \sqrt{\left(\rho C/L\right)U_{tip}}
\]

Rotating cavitation is a local cavitation instability related to rotating stall that occurs when a turbomachine operates at high incidence angles or at low cavitation numbers. At incidence angles near the value where the blade stalls, some blades may stall before others, as shown by the streamlines in Figure 1.8, and generate a wake cavity or “stall cell” and corresponding blockage. The flow blockage at one blade passage diverts fluid to another blade passage, resulting in an incidence change on the next blade, causing a stall cell formation on the next blade. In this way, a stall cell can propagate circumferentially at some fraction of the rotor rotation rate, blocking flow asymmetrically around the axis of rotation, and inducing transient forces on the rotor [6]. The propagation rate of rotating cavitation has been observed at both sub and super-synchronous frequencies, although the super-synchronous frequencies are more common [12]. Typical frequencies range from 10 – 20% higher than the pump rotation frequency [6]. Multiple stall cells can form, introducing greater forces and further blocking flow. In a turbopump, this rotating stall is sometimes referred to as rotating stall with cavitation [6]. Inducers that do not experience rotating stall may experience rotating cavitation when required to operate at low cavitation numbers. If one or two blades manifest a greater degree of cavitation, a cavitation volume functionally equal to the stall cell can form producing the same transient forces [6]. Rotating cavitation instabilities are generally formed by a local imbalance caused by an asymmetric cavity rotating from blade to blade, which causes a varying radial load and shaft orbits [1, 6, 10]. Some say rotating cavitation typically occurs when the length of a cavity approaches 65% of the blade to blade spacing [13], but others assert that cavities become unstable between 65 – 90% [11]. Pump failure occurs when the oscillatory loading essentially shakes the pump to failure. In this work, rotating stall with cavitation and rotating cavitation will be lumped together and simply referred to as rotating cavitation.
1.3 Secondary Flows

Flow phenomena contrary to the mean flow of the fluid in a turbomachine are often called secondary flows. In addition to cavitation, secondary flows can also have strong effects on inducer performance as well as influencing cavitation formation. The secondary flow that has the greatest influence on inducer performance is backflow [6], which is a region of flow upstream of the rotating turbomachinery that has a velocity component opposing the core flow. Backflow generally occurs at high incidence [6], most commonly caused by low flow coefficient. Figure 1.9 shows the backflow upstream propagation for the same inducer operating at two flow coefficients; the design flow coefficient (a), and 60% of the design flow coefficient (b). Blue regions in Fig. 1.9 identify flow with a velocity component opposite that of the core flow. Backflow extends \( \sim 0.3 \) tip diameters upstream when operating at the high flow coefficient, but increases to \( \sim 2.5 \) tip diameters upstream when flow coefficient decreases to 60% of design. Significant backflow at the inducer leading edge induces unstable pump conditions [14, 15], which affects the incidence angles and overall pump performance [6]. Backflow that penetrates far upstream of the inducer restricts the
upstream flow area, accelerating the inlet flow. This can lead to upstream cavitation, which grows and collapses with time [4, 16]. Figure 1.10 provides a contour plot of the volume fraction of vapor in an inducer experiencing upstream cavitation [4].

Figure 1.9: Typical backflow upstream propagation: (a) at high flow coefficient and (b) at low flow coefficient. Blue regions identify flow with a velocity component opposite that of the core flow.

Japikse’s two-elements-in-series (TEIS) model is a useful way to understand inducer inlet flow [17]. This is a 1D flow model where the inducer inlet from upstream to the throat is considered as a diffuser. The area ratio (AR) of this diffuser is the ratio of the blade throat area to the upstream inlet relative flow area at the tip section and is found using Eqn. 1.11 where $A_{throat}$ is the minimum area in a blade passage, and $A_{inlet}$ is the bundle of streamtubes in the relative reference frame upstream of the blade leading edge that must contract to pass through the throat [17, 18].

\[
AR = \frac{A_{throat}}{A_{inlet}} = \frac{\sin\beta_b}{\sin\beta} = \frac{\sin\beta_b}{\sin(\text{atan}\phi)}
\]  

(1.11)

The inlet area of the TEIS diffuser is proportional to $\sin(\beta)$ while the discharge area is proportional to $\sin(\beta_b)$ [17]. Recall the inlet flow angle can be approximated as $\tan^{-1}(\phi)$ for
flows with no upstream tangential velocity. The inlet blade angle is fixed at the blade tip. If the area ratio of the inlet diffuser in this TEIS system is too large then the inlet will stall [19]. Japikse showed that at a critical area ratio, $AR_{\text{crit}}$, above $\sim 1.5$ the inlet will start to stall, yielding backflow [19, 20]. The ideal pressure recovery of this inlet diffusing section can be expressed as $1 - 1/AR^2$; hence, as with all diffusers, higher area ratio means theoretically higher pressure recovery or higher static pressure. Inducers often operate close to this stall limit, because the high pressure recovery minimizes the growth of the vapor cavity and delays head breakdown [4].

In addition to the area ratio, the inducer inlet diffuser pressure recovery is another metric that can be used to predict if backflow will occur. Pressure recovery can be defined with Eq. 1.12, where $\eta_a$ is the diffuser effectiveness, which is a constant value used to describe the losses in a diffuser and helps match TEIS curve fits to experimental data. Diffuser effectiveness in Eq. 1.12 is typically between 0.5 and 0.85. Backflow has been found to form in inducers operating with a pressure recovery greater than 0.56 [19, 20].

$$c_p = \eta_a \left( 1 - \frac{1}{AR^2} \right) \quad (1.12)$$

Prior studies, both experimental and numerical, have explored the cause and behavior of backflow. Experimental work has shown that inlet reverse flow can occur near the casing wall due to a significant axial pressure gradient between the blades. This gradient causes the pressure rise to be produced in the first half of the inducer while leaving the latter half largely unloaded [21].
An axial momentum defect near the casing wall at the impeller inlet caused by the boundary layer also contributes to backflow formation [21], and is affected by tip clearance flow [22].

Computational studies have also explored the effect of tip clearance on backflow. Kim observed that backflow penetrates further upstream and performance declines rapidly for larger tip clearances, but offered no explanation of the cause [23]. Kim also found that for small tip clearances, backflow upstream propagation is reduced, but flow begins to separate at the hub in the middle of the passage, also reducing performance [23]. You et. al. showed that larger tip clearance creates a larger tip vortex and creates negative pressure regions along the tip vortex in a linear cascade [24].

These studies attribute backflow to tip leakage flow, driven by the pressure gradient across the blade tip. Figure 1.11, adapted from [6] illustrates how the pressure gradient between the pressure and suction sides of the blade causes flow to reverse direction and move back over the blade between the blade tips and pump casing. This flow generates a vortex structure near the leading edge of the blade [6] called the tip vortex. At flow coefficients below design, the blades are more heavily loaded, causing a more pronounced tip leakage flow [23].
1.4 Thermal Effects

Turbopumps are often used in spacecraft to transfer liquid propellants, including liquid hydrogen and oxygen. The critical temperatures of these fluids are very low, 33 K for hydrogen, and 154 K for oxygen. At the cryogenic working temperatures of these liquids, fluid properties, most notably vapor pressure, vary significantly with small changes in temperature. Figure 1.12 plots vapor pressure with temperature for liquid hydrogen and oxygen to visualize the strong dependence of vapor pressure on fluid temperature. Cavity formation introduces thermal effects, especially for liquid hydrogen, because of this. At cryogenic temperatures, the ratio of liquid to vapor density is low which requires a greater mass of liquid to vaporize to form a cavity. This emphasizes the effects of evaporative cooling, which reduces fluid temperature surrounding the bubble. A corresponding drop in vapor pressure in the fluid surrounding the cavity follows the drop in fluid temperature, effectively suppressing cavitation and improving pump performance [25]. This is known as thermodynamic suppression head (TSH) [6, 26, 27]. The effects of TSH on NPSH are defined using Eq. 1.13. TSH lowers \( \text{NPSH}_{\text{required}} \), which is desirable as the pump can then operate at a lower NPSH while still avoiding cavitation. At lower than design flow coefficients, the TSH is reduced by turbulent mixing induced by inducer backflow [28].

Figure 1.12: Vapor pressure of liquid hydrogen and liquid oxygen at various temperatures. Vapor pressure is highly dependent on fluid temperature for these liquids.
(NPSH)_{required} = (NPSH)_{ideal} - TSH \tag{1.13}

In addition to lowering NPSH_{required}, TSH can also affect the shape of a breakdown curve. Inducers pumping cold water often experience sharp and sudden breakdown, characterized by a sharp knee in the breakdown curve (like those in Fig. 2.6). This sharp breakdown is caused by the rapid formation of large, well-defined cavity volumes that can significantly reduce pump performance. In liquid hydrogen, cavity volumes are not as well-defined as those in cold water. Instead, cavity volumes in liquid hydrogen are porous, spread out, and in some cases can be described as a gradient. Because the ratio of liquid to vapor density of hydrogen can be much lower than that of water, the mass rate of evaporation to generate a cavity of a specific volume is much greater in liquid hydrogen than in cold water. Therefore, the heat which must be conducted to the liquid-vapor interface of the cavity is much larger. This causes the temperature in the cavity to fall below that of the bulk liquid, which causes a significant drop in the vapor pressure in the bubble, and limits the driving force behind bubble growth [6]. This restricts pump performance degradation due to cavitation formation which can lead to a more gradual breakdown, with a less pronounced knee for inducers pumping liquid hydrogen than for inducers pumping cold water.

Due to the complexity and cost of working with cryogenic fluids, much of the work done to understand the effects of cavitation on the performance of inducer systems designed for liquid rocket turbopumps has been performed with cold water as the working fluid. Cold water does not exhibit the thermal effects caused by significant changes in fluid properties with marginal temperature changes, and does not provide an accurate representation of inducer performance under real world cryogenic conditions. Understanding and accurately simulating the thermal effects of cavitation in cryogenic fluids is necessary for improved design of liquid rocket turbopumps.

Experimental studies exploring the thermal effects of cavitation and cavitation performance under cryogenic conditions have long been available [29–37], and attempts have been made to predict TSH effects for various fluids in pumps [37, 38]. An empirical derivation of TSH in liquid hydrogen as a function of temperature has been developed and is shown in Eq. 1.16, where $T_t$ is the total temperature [38]. While useful in determining the magnitude of TSH and the corresponding NPSH, this function provides no estimation of inducer head or stability. The standard way to
determine TSH is to use the NPSH to inlet velocity head ratio, defined in Eq. 1.15, where $Cm$ is the average meridional velocity. To determine TSH, let liquid hydrogen at 21 K or higher and $\phi < 0.15$ be defined to operate at a NPSH to inlet velocity head ratio of 1. A fluid with TSH can then operate at a NPSH to inlet head velocity ratio of 3, for the same conditions. The definition for TSH in liquid hydrogen inducers is then defined by Eq. 1.16 [39]. This method is only a rough estimator of TSH that does not consider fluid property changes with temperature and is only valid when the inlet velocity head is small relative to the vapor head. However, it is based on experimental data for liquid hydrogen around 21 K [39].

$$TSH_{LH_2} = 0.415(T_t - 20.0)^2$$

$$NPSH = \frac{Cm^2}{2g}$$

$$TSH_{LH_2} = \frac{Cm^2}{g}$$

It is only recently that efforts to simulate cryogenic cavitation using computational fluid dynamics (CFD) have been made [25, 40–44]. These efforts have resulted in a multi-element unstructured numerical framework called CRUNCH CFD [44]. This code solves the energy equation for a multiphase mixture in conjunction with conservation of mass and momentum, while considering the evaporative cooling effects of cavitation [40], and has been shown to slightly under predict cavitation performance of an inducer with liquid hydrogen as the working fluid [41]. However, considering the complexity of the problem and the sensitivity of the inducer to small changes in flow rate, the agreement between CRUNCH CFD and experimental results is reasonably good, and provides a much improved estimation of the resulting flow field when compared to typical cold water CFD simulations.

1.5 Stability Control Device

Three main factors limit the ability to design inducers that operate at low flow coefficients. First, small blade angles reduce throat area, which require thin blades that introduce structural
weaknesses. Second, low critical incidence angles reduce cavitation instabilities but limit flow range. Finally, the propensity for the inducer to stall and develop backflow below the design flow coefficient [4]. The current design approach often results in a limited operational flow range and unfavorable suction performance at lower than design flow coefficients. Multiple solutions to improve operational flow range and suction performance have been developed. A backflow deflector was developed in 1971 to limit backflow upstream penetration [45]. Casing modifications to sequester backflow to a small groove near the leading edge of the blade limited cavitation, but only near the design flow coefficient [46, 47]. Other systems employing a bleed slot and recirculation channel were developed [48–50], that greatly improved the operating range and stability. Another promising solution, and an improvement over prior designs came in the form of a stability control device, or SCD, invented by Japikse [51], and shown in Fig. 1.13.

The SCD employs a bleed slot downstream of the leading edge of the inducer blades to divert high energy fluid. This energized fluid continues into a radial diffuser, which captures energy in the blade tip vortex that would be otherwise lost and uses it to help reinject flow upstream of the inducer. The main difference between the earlier backflow recirculation devices and the SCD is the inclusion of a radial diffuser positioned after the bleed slot in an SCD. The flow is then reinjected into the core flow upstream of the inducer. As a result, the SCD resolves the design difficulties by regulating inducer incidence, and allows for larger blade angles and a larger
flow range while minimizing the risk of stall and backflow [4]. Other studies characterizing how varying SCD and inducer blade designs affect breakdown, both numerically and experimentally, have been performed and show improved stability and suction performance with reductions in rotordynamic forces at both on and off design flow coefficients. More precisely, Japikse and Baun showed that an SCD suppresses cavitation instabilities and increases suction performance in certain cases [52]. Oliphant showed that use of an SCD enables a high suction performance inducer to operate stably over a wide range of flow coefficients while minimizing failure rate, and increasing working life [4]. Krise showed that implementing an SCD can remove the region of backflow at the leading edge of a flat plate inducer under steady flow conditions [53]. Cluff developed a method to predict an important parameter called the SCD mass flow gain from inlet blade tip angle, and showed that efficiency and SCD mass flow gain are inversely related [54]. The SCD mass flow gain, also called the mass flow gain factor (K), is defined in Eq. 1.17, where $\dot{m}_{SCD}$ is the mass flow through the SCD, and describes the increased mass flow due to the additional flow reintroduced to the inlet tube by the SCD.

$$
K = \frac{\dot{m}_{inlet} + \dot{m}_{SCD}}{\dot{m}_{inlet}}
$$

Lundgreen showed that SCD surge instabilities are caused by a negative cavitation compliance occurring in the SCD bleed slot, and suggested three improved SCD design parameters. The mass flow rate through the SCD should be minimized but still sufficiently high to suppress backflow at the leading edge of the blade. Surge events in the SCD can be suppressed by removing the tangential velocity in the fluid passing through the SCD. Finally, Lundgreen suggests widening the SCD bleed slot. The width of the SCD bleed slot should be such that flow separation is prevented in the radial diffuser of the SCD [8]. In addition to the improved SCD design parameters, Lundgreen also stresses the importance of the mass flow gain factor in inducer and SCD design [8]. The performance improvements offered by SCD implementation are significant, but are dependent on good SCD design, or a SCD designed together with an inducer.
1.6 Contributions from the Present Work

Turbopump and inducer performance has been studied for years, both experimentally and computationally. Despite the wide range of work done to understand turbopump flow physics, the origins of backflow formation are still debated. While recent work has explored the effectiveness of SCD implementation with CFD, water was used as the working fluid. Most inducer performance is evaluated during steady operation, but inducers must accelerate from rest to reach the steady state. Experimental work has been done to describe the unsteady startup of inducers, but it is difficult to simulate inducer startup. Therefore, very little, if any, CFD work has been performed to model the startup of inducers. The work presented in this dissertation covers many aspects of inducer performance, but serves to advance the understanding of turbopump operation through CFD simulations.

This dissertation strives to meet the following objectives by simulating inducer performance during a variety of operational conditions:

1. Determine the impact of tip leakage flow on backflow formation.

2. Improve the current understanding of the effects of SCD implementation with an inducer pumping liquid hydrogen.

3. Determine how cavitation formation affects the performance of an inducer during transient start-up.

Through analysis of data obtained from extensive CFD simulations, a more complete understanding of inducer performance and flow physics is obtained.

1.7 Dissertation Organization

The chapters of this dissertation are written for publication, and as such, most of the following chapters contain an introduction, a literature review, a methods section, a presentation and discussion of results, and some conclusions. Chapter 2 is a methodology section that describes how the work in the following chapters was performed. Chapter 3 is a paper published in the Journal of Fluids Engineering [55], that addresses misconceptions about the effects of tip leakage...
flow on backflow formation, and solidifies inlet diffusion to be the flow phenomenon that drives backflow upstream propagation. Chapter 4 is a paper to be published that evaluates inducer performance with a SCD when the working fluid is liquid hydrogen. Chapter 5 describes a simulation method to model the unsteady start-up of a centrifugal impeller. Chapter 6 is a conclusions chapter summarizing the body of work completed for this dissertation.
CHAPTER 2. METHODOLOGY

This dissertation presents results of simulations using computational fluid dynamics (CFD) to model impeller flow fields and predict the performance of various impeller geometries operating at multiple flow conditions with and without a SCD. Two CFD packages were used, Star-CCM+ version 11.04.010 and CRUNCH CFD version 3.0.0.

2.1 Star-CCM+

Star-CCM+ is a commercial CFD software package that is relatively easy to use and offers a broad range of simulation capabilities [56]. The work presented in this dissertation employs multiple simulation schemes in Star-CCM+ to simulate various aspects of inducer performance.

2.1.1 Meshing

The cornerstone of any CFD simulation is the mesh. A mesh is the discretized representation of a geometric domain [57]. The quality of the mesh has a very strong influence on the quality of the results. All simulations completed in Star-CCM+ for this work are three-dimensional finite volume internal flows and range from $\sim 9$ million cells to $\sim 16$ million cells. These meshes were generated using the meshing package built in to Star-CCM+. For finite volume simulations, solutions to governing equations are found at the center of mesh cells. These mesh cells vary between tetrahedral, polyhedral, hexahedral, or wedge volumes throughout the computational domain. Figure 2.1 shows a representative tetrahedral cell and a representative polyhedral cell. Polyhedral cells generated by Star-CCM+ typically have an average of 14 cell faces. For all simulations performed using Star-CCM+ presented here, the computational domain was initially divided into tetrahedral cells from the input geometry, resulting in an unstructured mesh. Tetrahedral cells provide an efficient and simple solution for complex meshing, but allow for limited stretching and poor gradient approximation [57]. A dualization scheme is then used to mark the center of the tetrahedral cells.
and midpoints of boundary edges. Polyhedral cells are then generated starting from the boundary edges and using the center of the tetrahedral cells. Polyhedral cells provide a balanced solution, and do not have the shortcomings of tetrahedral cells. Gradients are better approximated because of the increased number of neighboring cells, and are less sensitive to stretching. Practical tests have shown that converting a tetrahedral mesh to a polyhedral mesh reduces the number of cells by $\sim 4 - 5$ times, memory requirements are approximately halved, and required computing time is between a tenth to a fifth with the same accuracy of the tetrahedral mesh [57]. Hexahedral cells are not as flexible as polyhedral cells, and are typically used in structured meshes. Cross-sections of an unstructured mesh and a structured mesh are shown in Fig. 2.2 to illustrate the difference between hexahedral and polyhedral cells. Meshes consisting of hexahedral cells typically contains more cells than a mesh of polyhedral cells, but less than a mesh of tetrahedral cells, with a similar comparison in required computational times [58, 59].

While structured meshes consist mostly of hexahedral cells, unstructured meshes can use hexahedral cells near the wall boundaries to resolve the boundary layer. Figure 2.3 shows a detail image of a cross-section of a typical unstructured mesh to highlight the prism layer cells. The prism layer is comprised of a series of hexahedral cells that grow in size with increasing distance from the wall. The size of the cell closest to the wall is important in mesh generation as this defines $y^+$. The dimensionless wall distance, $y^+$, is defined in Eq. 2.1, where $\Delta y$ is the distance of the near-wall node to the wall, $\nu$ is the local kinematic viscosity of the fluid, $\tau_w$ is the wall shear stress, and $\rho$ is the fluid density. If the $y^+$ is too large, the boundary layer will not be properly resolved by the mesh alone. There is no specific recommended $y^+$ that applies to all simulation setups. However, low $y^+$ values will provide a more accurate prediction of boundary layer velocity, temperature and
Figure 2.2: (a) Cross-section of typical structured mesh with hexahedral cells. (b) Cross-section of typical unstructured mesh with polyhedral cells.
separation. While low $y^+$ values improve solution accuracy, they increase overall cell count and computational cost. The average $y^+$ for meshes presented here that were made using Star-CCM+ range between $\sim 0.1 < y^+ < \sim 75$. While the higher range of $y^+$ values does not correspond to the ideal mesh density, it is necessary to maintain a balance between solution accuracy and computational cost. To offset the relative mesh coarseness, wall functions are employed for the meshes with a $y^+ > 30$ to better resolve the boundary layer. More specifically, the two-layer all-$y^+$ model uses algebraic relations based on the assumed velocity, temperature, and turbulence distributions across the boundary layer to provide valid boundary conditions for flow, energy and turbulence quantities for a wide range of near-wall mesh densities [57]. This allows for a little more flexibility in mesh density while still providing a reasonably accurate solution.

$$y^+ = \frac{\Delta y}{v} \sqrt{\frac{\tau_w}{\rho}}$$

Three different meshes made in Star-CCM+ are used in this dissertation. Chapter 3 explores the flow differences between an inducer with and without tip clearance. The only difference between the two geometries is that the shroud radial position is reduced to be the same as the blade tip, resulting in zero tip clearance for one inducer to prevent tip leakage flow. The mesh with tip clearance consists of 6.8 million cells, where 99.8% are polyhedral cells, 0.18% are hexahedral
cells, 0.001% are wedge cells, and 0.0001% are tetrahedral cells. The mesh with the reduced radius shroud resulting in no tip clearance consists of 5.7 million cells, where 99.8% are polyhedral cells, 0.085% are hexahedral cells, 0.034% are wedge cells, and 0.00003% are tetrahedral cells.

Chapter 5 explores the flow during fast start-up of an inducer. There is only one mesh considered for this case, and is comprised of 16.3 million cells, consisting of 99.8% polyhedral cells, 0.13% hexahedral cells, 0.12% wedge cells, and 0.003% tetrahedral cells.

### 2.1.2 Time Discretization

Once suitable meshes have been generated, the desired time discretization scheme must be considered. As cavitation formation and growth can be time-dependent, all simulations attempting to model inducer cavitating performance must be time-accurate, and run with an unsteady solver. As the considered flow fields are complex, it is difficult to reach a converged solution with an unsteady solver from constant, arbitrary initial conditions. Therefore, inducer flow fields are initially simulated without cavitation with a steady solver. The pressure and velocity results from this simulation are then exported and used as initial conditions for a cavitating simulation with an unsteady solver using first-order temporal discretization. The rotation rate of the inducer is one contributing factor to determine the required time step for an unsteady solver. Time steps selected to result in $\sim 0.5^\circ$ of rotation per time step has been shown to accurately resolve the flow and cavitation formation [8]. Additional time step information specific to each chapter is provided in Chapter 3, 4, and 5.

### 2.1.3 Time and Grid Independence

The inducer geometry considered in Chapter 3 was also considered by Lundgreen, who performed time and grid independence studies [8]. He considered time steps of $1.2 \times 10^{-4}$ s, $2.8 \times 10^{-4}$ s, and $1.4 \times 10^{-5}$ s, and found a large mass flow imbalance occurred at the largest time step, but not the lowest. He reported the percent change of cavitation number and head coefficient with changing time step. Those results are reported here in Table 2.1. Based on these results, the time step for all cavitating simulations in Chapter 3 were defined to result in $0.5^\circ$ of rotation per time step.
Table 2.1: Inducer performance for converged solutions with varying time steps for the 7° inducer considered in Chapter 3, at $\phi = 0.07$ and $\sigma \approx 0.022$. Percent changes are calculated from the values corresponding to the time step of $1.4 \times 10^{-5}$ s. From [8].

<table>
<thead>
<tr>
<th>Time Step (s)</th>
<th>$\sigma$</th>
<th>$\psi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1.2 \times 10^{-4}$</td>
<td>$-1.93%$</td>
<td>$-0.26%$</td>
</tr>
<tr>
<td>$2.8 \times 10^{-4}$</td>
<td>$-1.34%$</td>
<td>$-0.74%$</td>
</tr>
<tr>
<td>$1.4 \times 10^{-5}$</td>
<td>$0.022$</td>
<td>$0.27$</td>
</tr>
</tbody>
</table>

Lundgreen also performed a grid independence study and increased mesh size from 6 million cells to 12 million cells. He found the cavitation number, head coefficient, and mass flow monitors varied less than 0.5% from the 6 million cell mesh values [8]. Therefore, the meshes in Chapter 3 are considered to be sufficiently refined to capture the flow physics. Similar mesh and time procedures were used in the formulation of the simulations in Chapter 5 and should be sufficient to accurately model the considered flow.

2.1.4 Rotation

Rotation can be modeled in two ways with Star-CCM+, using a rotating reference frame or rigid body motion [57]. Rigid body motion moves the mesh cell vertices a fixed displacement per time step, and must be employed in a time-accurate analysis. While this method is the most accurate approach to simulate impeller rotation, it can be very computationally expensive for large meshes [57]. While less accurate, a rotating reference frame is less computationally intensive and provides a compromise between simulation accuracy and computational cost. For this reason, all Star-CCM+ simulations presented here employ a rotating reference frame to simulate rotation. The rotor surfaces, which are defined as walls with the no slip boundary condition, are the only parts defined to be in the rotating reference frame, and are the only parts subject to rotation in the computational domain. All other boundaries in the domain are defined to be in the lab reference frame, which is stationary.
2.1.5 Cavitation Modeling

While a simple constant density equation of state model is used to model single phase flow, Star-CCM+ provides three cavitation models [57]. For the work presented here, the Rayleigh-Plesset equation is used to determine the cavitation bubble growth rate. An Eulerian multiphase volume of fluid method [60] is used to describe cavitation formation and flow with liquid water as the primary phase and water vapor as the secondary phase. Liquid water is defined by a vapor pressure of 3170 Pa with a density of 998.2 kg/m$^3$, to simulate water at 25° C. This cavitation modeling method solves the single phase governing equation set for an equivalent fluid with physical properties defined as functions of constituent phases and volume fractions.

A Rayleigh-Plesset formulation that includes the influence of bubble growth acceleration along with viscous and surface tension effects is employed to model rate of vapor production. This is the most detailed model available in Star-CCM+ to simulate cavitation. Other models are based on a simplified Rayleigh-Plesset formulation or a finite rate equation. These simplified models neglect the influence of bubble growth acceleration, surface tension effects, and viscous effects. While the full Rayleigh-Plesset formulation used for the simulations presented here is more accurate, it can require additional mesh and time resolution, which increases computational cost and increased numerical error potential. Additionally, research suggests that viscous and surface tension effects have little effect on simulation accuracy [61]. While the full and reduced Rayleigh-Plesset models can both be used to model cavitation for a wide range of applications, they do not perform well in thermal non-equilibrium.

In the volume of fluid model, a single set of momentum and turbulence equations are solved to find the distribution of the continuous phase. The dispersed phase is then modeled with a transport equation for the volume fraction [60]. The density and dynamic viscosity are calculated as functions of the physical properties of the constituent phase and its volume fraction. This method of cavitation modeling is used for all Star-CCM+ cavitating simulations.

Breakdown curves are used to describe inducer cavitating performance. Using a converged single phase solution as initial conditions for a cavitating simulation, the static pressure at the pump outlet is controlled to change the pump inlet pressure and cavitation number. To generate a breakdown curve, outlet pressure is initially set to be high to approximate the single phase solution
to aid convergence. After the solution has converged, outlet pressure is progressively decreased to
determine performance at lower and lower cavitation numbers.

### 2.1.6 Turbulence Modeling

For all Star-CCM+ simulations performed for this work, the realizable k-\(\varepsilon\) model was used
to model turbulence. This model offers mesh flexibility by applying wall functions to model the
boundary layer in the viscous sublayer if the wall \(y^+ > 30\), and assumes the mesh density properly
resolves the viscous sublayer for regions where the wall \(y^+ < 30\). Despite the relatively large
overall mesh sizes, the average \(y^+\) of some of the meshes presented here are not necessarily ideal,
and wall functions are applied by the turbulence model to help offset the relative coarseness of the
mesh to properly resolve the boundary layer.

Lundgreen modeled the flow physics of a similar computational setup of an axial inducer
with a 7° tip blade angle during quasi-steady operation using the same turbulence and cavitation
models with good agreement when compared to experimental data, predicting the cavitation num-
ber where head breakdown occurs within 1% of the experimentally observed value [16].

### 2.1.7 Boundary Conditions

The meshes in Chapter 3 consist of a single computational region with boundary conditions
applied to each geometric surface. Figure 2.4 shows the computational domain for simulations
presented in Chapter 3 and distinguishes the boundary conditions. The rotor and rotor hub, which
are defined as no slip walls, are the only parts defined to be in the rotating reference frame, and are
the only parts subject to rotation in the computational domain. The rotor shroud is also defined as a
no slip wall, but is defined to be in the lab reference frame, which is stationary. The inlet boundary
is defined as a mass flow inlet, which provides a constant mass flow with time at all radial locations,
and is used to control pump flow coefficient. The outlet boundary is defined as a pressure outlet,
and dictates the static pressure on the outlet surface. The pressure outlet defines a radial equilibrium
of pressure, which is suitable for the strong rotational flows of an axial inducer [57]. In a radial
equilibrium pressure outlet, the specified static pressure is only applied at the minimum radius of
the boundary. The static pressure at all other radial locations is defined by the relationship in Eq.
2.2, where \( r \) is the radial location, \( r_i \) is the minimum radius relative to the axis of rotation, and \( V_\theta \) is the circumferential average of tangential velocity.

\[
P_s = P_{s,\text{hub,def}} + \int_{r_i}^{r} \frac{\rho V_\theta^2}{r} dr
\]

The boundary conditions for the mesh in Chapter 5 are very similar to those in Chapter 3, but instead of one computational region, the mesh in Chapter 5 is comprised of two computational regions joined by an internal interface separating the region around the inducer from the upstream inlet tube. Figure 2.5 shows the computational domain for the simulations presented in Chapter 5 and distinguishes the boundary conditions. Again, the rotor surfaces are defined to be in the rotating reference frame while all other surfaces are in the global, non rotating reference frame. The inlet and outlet boundaries are again defined by a mass flow inlet and a pressure outlet, respectively. However, additional modifications made to the outlet boundary pressure definition are described in detail in Chapter 5.

2.1.8 Validation

Comparing simulation results to experimental data is the best way to validate the selected modeling parameters. Concepts NREC provided Lundgreen with experimental data for the 7°
inducer used in Chapter 3 for this work [8]. Lundgreen used the same simulation parameters and setup in his work that is used for the inducer simulated in Chapter 3. Fig. 2.6 provides Lundgreen’s comparison of simulated cavitating performance to experimental data [8]. When comparing his simulation data to experimental data, Lundgreen found good agreement between the two, with the CFD simulation under predicting head coefficient by 20% between $0.02 < \sigma < 0.05$ for the inducer with no SCD operating at $\phi = 0.042$. This under prediction was caused by significant cavitation instabilities. At all other operating conditions for all considered geometries, Lundgreen shows good agreement between CFD and experimental results [8], with average error between CFD and experimental results of less than 3% above the breakdown point, and average error of $\sim 15\%$ below the breakdown point. The simulations presented here in Chapter 3 were performed the same way Lundgreen performed his simulations, and his validation can therefore be applied here.

The simulations presented in Chapter 5 are significantly different than those in Chaper 3, and model the transient start-up of a centrifugal impeller. While inducer operation under these conditions have been explored experimentally by others [62–71], and significant effort was made to obtain start-up data for this work, no experimental data were acquired. No validation is presented for the simulations of Chapter 5 due to lack of experimental data.
Figure 2.6: Comparison of simulated cavitating performance to experimental data. From [8]. (a) 7° inducer with no SCD. (b) 7° inducer with an SCD. (c) 9° inducer with an SCD.
2.2 CRUNCH CFD

CRUNCH CFD is very different than Star-CCM+. CRUNCH is a more specialized unstructured grid solver and offers advanced models for turbulence, cavitation, combustion, and cryogenics. While CRUNCH does offer a GUI, it is not as intuitive as that of Star-CCM+ and it was more efficient to operate CRUNCH from the command line. Additionally, CRUNCH offers no meshing or post processing capabilities [72]. As such, all the meshes presented in Chapter 4 were generated using Autogrid [73], and any post processing was performed using Paraview [74]. The work presented in this dissertation employs multiple simulation schemes in CRUNCH CFD to compare cryogenic performance of an inducer with and without a SCD.

2.2.1 Meshing

As CRUNCH has no meshing capabilities, all meshes in Chapter 4 were generated using Autogrid5, and are structured. These structured meshes were imported to CRUNCH and converted to unstructured meshes, but retain the hexahedral cell composition. Figure 2.7 shows cross sections of the full no SCD and SCD meshes used in Chapter 4 with CRUNCH. Note the SCD restriction near the reinjection slot in Fig. 2.7. This restricting is meant to control flow velocity in the SCD in an attempt to prevent cavitation growth and surge in the SCD, which can negate the benefits of SCD implementation [8]. The mesh defining the inducer with no SCD is comprised of $9.5 \times 10^6$ hexahedral cells, while the mesh defining the inducer with SCD is comprised of $12.1 \times 10^6$ hexahedral cells. The cell count differences between the two meshes stems from the additional computational domain added by the SCD volume, and the corresponding refinement near the SCD bleed and reinjection slots. Detailed images of the blade tips are provided in Fig. 2.8 illustrating the increased refinement in the SCD mesh between the blade tips and the shroud. The inducer with no SCD mesh uses 6 cells to span the tip gap, and has an average $y^+$ of 68. The inducer with SCD mesh uses 14 cells to span the tip gap and has an average $y^+$ of 55. This refinement was required to allow for minimal mass flow imbalance across the interface to the SCD volume, and the internal interface separating the inducer region from the SCD region was extended $0.4 \times D_{tip}$ upstream and downstream of the leading edge of the inducer to further prevent mass flow imbalances across the interface. The interface between the inducer and SCD regions is detailed in Fig. 2.9, which shows
both upstream and downstream limits of the interface. Multiple interface locations, sizes, and cell counts were tested, but the interface configuration shown in Fig. 2.9 was found to cause the least mass flow imbalance across the interface and between the pump inlet and outlet. While the $y^+$ values for these meshes are not ideal, these meshes are an improvement over previous simulations and this work contains some of the most advanced published simulations with an SCD.

Figure 2.7: (a) Cross-section of CRUNCH mesh with no SCD. (b) Cross-section of CRUNCH mesh with SCD.
2.2.2 Time Discretization

While CRUNCH does support time-accurate analysis, all simulations presented in Chapter 4 were performed with a steady solver. Unsteady solvers are ideal for simulating cavitating performance and would have a better chance of accurately predicting pump performance, but the unsteady model in CRUNCH is significantly more difficult to use. All simulations would increase drastically in engineering and computational time and training. Because so many simulations are
required to generate a breakdown curve to describe cavitating inducer performance, the additional
time and computational cost of running unsteady simulations was deemed impractical. Fortunately
experimental results are readily available for the inducer considered in Chapter 4, and the steady
simulation results provide a reasonable approximation of the resulting flow field and are a signifi-
cant research contribution. These results are more thoroughly discussed in section 2.2.3.
2.2.3 Grid Independence and Validation

The inducer considered in Chapter 4 has been thoroughly tested and experimental data was available [29]. Multiple meshes of varying density were generated to compare simulation to experimental data to describe the quality of the mesh as well as the accuracy of the CRUNCH simulations. Mesh density was increased to 16 million cells and reduced to 1 million cells from the original 9 million cell mesh. Figure 2.10 shows detail images of a cross section near the blade tips for the 16 and 1 million cell meshes to provide a visual representation of how mesh density changes. The same image for the 9 million cell mesh is shown in Fig. 2.8a.

The effects of mesh density changes on simulation results are shown in Figure 2.11, which shows the experimental data compared to the predicted head coefficients of the 1 million cell mesh, a 9 million cell mesh, and a 16 million cell mesh. The inducer cavitating performance varies significantly with mesh density. In general, the 9 million cell mesh predicts the highest head coefficient, the 1 million cell mesh predicts the lowest head coefficient, and the 16 million cell mesh falls in between. Above a NPSH of 300, the 16 million cell mesh predicts a 3.26% lower head rise than the 9 million cell mesh, while the 1 million cell mesh predicts a 13.4% lower head coefficient than the 9 million cell mesh. For $200 < \text{NPSH} < 300$, the 16 million cell mesh predicts a 4.2% lower head rise than the 9 million cell mesh. The difference decreases for $130 < \text{NPSH} < 200$, for the 16 million cell mesh, which predicts a 0.32% lower head rise than the 9 million cell mesh. For $130 < \text{NPSH} < 200$, the 1 million cell mesh predicts up to a 17.7% lower head rise than the 9 million cell mesh. At $\text{NPSH} \sim 100$, the 16 million cell mesh predicts a 0.8% lower head coefficient than the 9 million cell mesh. While actual difference between predicted head coefficient between the 9 and 16 million cell meshes is less than 5%, Fig. 2.11 exaggerates the difference due to the small head coefficient magnitudes. It should be noted that the 16 million cell breakdown curve lies between the 1 and 9 million cell breakdown curves. This is not ideal, and is likely caused by unnecessary mesh refinement in specific regions of the domain. If the mesh density was further increased around the inducer while the upstream mesh density was unchanged or even decreased, it would likely improve the results. However, as the solution varies by less than 5% when the mesh density is almost doubled, it seems the mesh refinement for results presented in Chapter 4 is acceptable, and can produce reasonable results.
To more accurately define grid independence, a Richardson based error estimation using a safety factor of 1.25 [75, 76] was performed to determine the grid convergence index (GCI). This information is presented in Chapter 4.

It is clear from Fig. 2.11 that the 9 million cell mesh most closely approximates the experimental data, shown by a black line. Above a NPSH of 300, the 9 million cell mesh under predicted head coefficient by 1.9% when compared to the experimental data. As NPSH decreases, the difference between simulated and experimental head coefficient increases to 39% at a NPSH...
Figure 2.11: Cavitating performance curves comparing experimental data to simulations generated with multiple mesh sizes

of 101. This large difference between experimental and simulated head coefficient a function of an over prediction of the NPSH at which breakdown occurs, rather than an incorrect estimation of head coefficient. If the complete simulated breakdown curve was shifted by an NPSH reduction of 50 ft, the curves vary by 2.6% on average through the full NPSH range, as shown in Fig. 2.12. While the unshifted CRUNCH results vary from experimental data, the trends are very similar and the variance is due to a simple over prediction of breakdown NPSH.

2.2.4 Rotation

Rotation modeling in CRUNCH is not as advanced as Star-CCM+. Currently, rotation can only be modeled in CRUNCH using a rotating reference frame with rotation only around the x-axis [72]. Additionally, for Star-CCM+ simulations, the rotor is defined to be in the rotating reference frame, but CRUNCH requires the opposite definition. In CRUNCH, the shroud, SCD walls and all other non inducer surfaces are defined to be in the rotating reference frame, while the inducer blades and hub are defined to be in the inertial reference frame. This method of definition
2.2.5 Cavitation Modeling

CRUNCH offers two cavitation models. The first is a simple finite rate model which does not require any additional transport equations. The second model is more sophisticated model where the cavitation cloud is assumed to be composed of bubbles, and an additional conservation equation describing the bubble surface areas is considered. Both models have been found to produce very similar results in the absence of large-scale unsteadiness [72]. When the working fluid is cryogenic, both models determine local vapor pressure from a saturation property database using the fluid local temperature. All CRUNCH simulations presented here use the simple finite rate model for cavitation for simplicity and to decrease computational costs.

Breakdown curves are again used here to describe inducer cavitating performance. Using a converged single phase solution as initial conditions for a cavitating simulation, the static pressure
at the pump outlet is controlled to change pump inlet pressure and NPSH. To generate a breakdown curve, outlet pressure is initially set to be high to approximate the single phase solution to aid convergence. After the solution has converged, outlet pressure is progressively decreased to determine performance at lower and lower cavitation numbers.

The CRUNCH finite rate cavitation model uses a cavitation source term to define cavitation formation. This cavitation source term is defined in Eq. 2.3, where \( K_f \) is the rate constant for vapor generation, \( K_b \) is the rate constant for reconversion of vapor to liquid. These rate constants are defined in Eqs. 2.4 and 2.5, where \( \tau_f \) is the time constant for vapor formation, \( \tau_b \) is the time constant for liquid reconversion, and \( L \) is a reference length.

\[
m_t = K_f \rho_L \phi_L + K_b \rho_g \phi_g \tag{2.3}
\]

\[
K_f = \begin{cases} 
0 & P < P_v \\
\frac{1}{\tau_f} \left( \frac{V_\infty}{L_\infty} \right) \left( \frac{P - P_v}{1/2 \rho_\infty V_\infty^2} \right) & P > P_v
\end{cases} \tag{2.4}
\]

\[
K_b = \begin{cases} 
0 & P < P_v \\
\frac{1}{\tau_b} \left( \frac{V_\infty}{L_\infty} \right) \left( \frac{P - P_v}{1/2 \rho_\infty V_\infty^2} \right) & P > P_v
\end{cases} \tag{2.5}
\]

The time constants \( \tau_f \) and \( \tau_b \) can have significant effects on the final simulation results, as they affect how CRUNCH models cavitation growth. These time constants are modified using \texttt{CAV_FWD} and \texttt{CAV_BACK} flags in the CRUNCH input file. These, and other flags will be discussed in more detail in Section 2.2.8. Smaller time constant values result in larger net rate constants \( K_f \) and \( K_b \) and vice-versa. Figure 2.13 shows how varying cavitation time constant affects the cavitating performance curve for a single geometry operating at a single flow coefficient. These curves were generated using the same inducer considered in Chapter 4 operating at 20.3 K at \( \phi = 0.101 \). Experimental data was taken from Meng [29]. Time constants were varied between 500 and 1000 for \( \tau_f \) and 750 and 2500 for \( \tau_b \). As can be seen in Fig. 2.13, most of the variation occurs for NPSH \( > 150 \), as cavitation growth increases drastically below this NPSH. Cavitation time constants had little effect on the predicted cavitating performance for NPSH \( > 150 \), as little cavitation occurs. However, Case 1 deviates from the majority of cases for NPSH \( > 150 \), and
under predicted $\psi$ by up to 5%. Similar deviation at high NPSH is evident in Case 7, which under predicted $\psi$ by up to 6% for NPSH $> 150$. The remaining cases all predict approximately the same values and vary by $\sim 2\%$ from the experimental data above a NPSH of 150.

As the inducer begins to break down due to cavitation formation, much more variability with cavitation time constants is evident. Case 1 again under predicted head coefficient for NPSH $< 150$, up to 13%. In addition to the under prediction of head coefficient, Case 1 does not predict the sharp knee corresponding to breakdown at NPSH $\approx 115$. Cases 2, 3, and 4 all over predict the breakdown NPSH by $\sim 32\%$, but as cavitation continues to grow and the inducer drops deeper into breakdown, Cases 2, 3, and 4 under predicted NPSH by $\sim 30\%$. Case 5 is similar to Cases 2, 3, and 4, but under predicted NPSH deep into breakdown by 43%. Cases 6 and 7 are similar to each other and tend to flatten out the breakdown curve like Case 1. Cases 6 and 7 do not exhibit the sharp breakdown characteristic of the experimental data, and when the inducer is operating deep in breakdown, they under predicted NPSH by $\sim 39\%$. It is clear from Fig. 2.13 that Cases 3, and 4 most closely approximate the experimental data, and do not vary significantly from each other.
Therefore for the CRUNCH simulations presented here, cavitation time constants corresponding to Case 4 are used; namely $\tau_f = 750$ and $\tau_b = 1500$.

While the CRUNCH simulation data under predicted NPSH deep into breakdown in Fig. 2.13, it over predicts NPSH at similar conditions in Fig. 2.11 despite using the same $\tau_f$ and $\tau_b$. This is most likely a function of flow coefficient and temperature differences. Fig. 2.13 provides simulation data for an inducer operating at $\phi = 0.101$ and 20.3 K, while Fig. 2.11 provides simulation data for an inducer operating at $\phi = 0.107$ and 21.1 K. The higher flow coefficient and temperature of Fig. 2.11 helps reduce cavitation as incidence angles are lower and vapor pressure is higher. The relatively large effects of small variances in flow coefficient and temperature on the predicted breakdown curve helps illustrate the difficulty of developing a model to predict inducer cavitating breakdown at cryogenic temperatures.

2.2.6 Turbulence Modeling

For all CRUNCH simulations performed for this work, a two equation k-$\varepsilon$ model was used to model turbulence. This model solves transport equations for the turbulent kinetic energy, $k$, and the turbulent dissipation rate, $\varepsilon$, in addition to the momentum and energy equations in the Navier-Stokes equations. Equation 2.6 shows the turbulent kinetic transport equation, where $\sigma_k$ is a modeling constant. Equation 2.7 shows the turbulent dissipation rate transport equation, where $\sigma_\varepsilon$, $C_1$, $C_2$ are modeling constants, and $f_1$ and $f_2$ are empirical modeling function to account for low Reynolds numbers near walls [72]. For high Reynolds numbers, $f_1$ and $f_2$ equal 1.

$$\frac{\partial \rho_m k}{\partial t} + \frac{\partial}{\partial x_i} \left[ \rho_m u_i k - \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] = P_k - \rho \varepsilon + S_k$$ (2.6)

$$\frac{\partial \rho_m \varepsilon}{\partial t} + \frac{\partial}{\partial x_i} \left[ \rho_m u_i \varepsilon - \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] = \left( C_1 f_1 P_k - C_2 f_2 \rho \varepsilon + S_\varepsilon \right) \left( \frac{\varepsilon}{k} \right)$$ (2.7)

CRUNCH also offers wall functions to improve results despite mesh coarseness at a wall, but may not be as accurate when flow separates [72]. The specific wall function model used in the simulations presented here is a variant of the So-Zhang Near-wall damping model [77], with a modified damping term where the coefficients are defined in Eqs. 2.8 - 2.13 [78].
\[ f_\mu = (1 + 4Re_T^{-3/4}) \tanh \left( \frac{Re_k}{125} \right) \]  

\[ f_1 = 1 - \exp \left[ - \left( \frac{Re_T}{40} \right)^2 \right] - \frac{0.24}{\cosh \left( \log \left( \frac{Re_k}{100} \right) \right)} \]  

\[ f_2 = 1 - \frac{2}{9} \exp \left[ - \left( \frac{Re_k}{1} \right)^2 \right] \]  

\[ S_e = \frac{14}{9} C_2 \mu_2 \left[ \left( \frac{\partial k^{1/2}}{\partial x} \right)^2 + \left( \frac{\partial k^{1/2}}{\partial y} \right)^2 + \left( \frac{\partial k^{1/2}}{\partial z} \right)^2 \right] \]  

\[ Re_T = \frac{\rho k^2}{\mu L^2} \]  

\[ Re_k = \frac{\rho \sqrt{k_y}}{\mu} \]

This wall function model in CRUNCH does not adaptively activate based on wall \( y^+ \) like the wall functions in Star-CCM+. Instead, if the user activates wall functions, they are always active near viscous walls for the simulations presented here. As with Star-CCM+, the CRUNCH meshes are relatively large, but the average \( y^+ \) of some of the meshes presented here are not necessarily ideal, and wall functions are applied by the turbulence model to help offset the relative coarseness of the mesh to properly resolve the boundary layer.

### 2.2.7 Boundary Conditions

The meshes in Chapter 4 consist of a single computational region with boundary conditions dictating the purpose of each geometric surface. Figure 2.14 shows the CRUNCH computational domain and labels the boundaries. The rotor blades and rotor hub are defined as viscous walls in CRUNCH which exhibit the no slip condition. These walls are defined to be in the inertial reference frame, and are the only stationary parts in the computational domain. The rotor shroud is also defined as a viscous wall, but is defined to be in the rotating reference frame, and rotates in a direction opposite that of a physical rotor. This simulates rotor rotation without having the rotor
rotate to reduce computational costs. The rotor shroud is split upstream of the leading edge of the inducer blades. The upstream shroud is defined to be a slip wall, as is the upstream rotor hub. These surfaces do not exhibit the no slip condition to prevent shear driven fluid rotation upstream of the inducer to avoid potential solution divergence. The inlet boundary is defined as a mass flow inlet, which provides a constant mass flow with time at all radial locations, and is used to control the pump flow coefficient as in Star-CCM+. The outlet boundary is defined as a pressure outlet, and dictates the static pressure on the outlet surface. The pressure outlet can be used to fluctuate outlet pressure, but all CRUNCH simulations presented here were defined by a constant outlet pressure.

### 2.2.8 Running

Appendix A provides a tutorial to develop and run a typical CRUNCH case. While all the information in Appendix A is available in the CRUNCH user manual and tutorials [72, 79], it provides a simplified process for a real usage example for simulating a rotating inducer pumping
cryogenic propellant operating under single phase and cavitating conditions without attempting to explain the full capabilities of CRUNCH.
CHAPTER 3. CONTRIBUTIONS OF TIP LEAKAGE AND INLET DIFFUSION ON INDUCER BACKFLOW

This chapter is a paper published in the Journal of Fluids Engineering of the American Society of Engineers [55]. The formatting of this paper has been modified to meet the stylistic requirements of this dissertation. Some additional details have been added as a result of discussion during the dissertation defense.

3.1 Contributing Authors and Affiliations

Tate Fanning, Steven Gorrell, Daniel Maynes, Department of Mechanical Engineering, Brigham Young University, Provo, UT 84602, USA. Kerry Oliphant, Concepts NREC, White River Junction, Vermont 05001, USA.

3.2 Abstract

Inducers are used as a first stage in pumps to minimize cavitation and allow the pump to operate at lower inlet head conditions. Inlet flow recirculation or backflow in the inducer occurs at low flow conditions and can lead to instabilities and cavitation-induced head breakdown. Backflow of an inducer with a tip clearance of $\tau = 0.32\%$, and with no tip clearance is examined with a series of CFD simulations. Removing the tip clearance eliminates tip leakage flow; however, backflow is still observed. In fact, the no tip clearance case showed a 37% increase in the length of the upstream backflow penetration. Tip leakage flow does instigate a smaller secondary leading edge tip vortex that is separate from the much larger backflow structure. A comprehensive analysis of these simulations suggests that blade inlet diffusion, not tip leakage flow, is the fundamental mechanism leading to the formation of backflow.
3.3 Introduction

An inducer is an axial rotor that imparts increased pressures and tangential velocities to a fluid and is often placed immediately upstream of an impeller. The purpose of an inducer is to improve pump suction performance, widen the operating range, and reduce cavitation throughout the machine. However, inducer effectiveness can be reduced by secondary flow formation. The secondary flow that has the greatest influence on inducer performance is backflow [6], which is a region of flow upstream of the rotating elements that has a velocity component opposing the core flow. This paper investigates the formation of backflow in an inducer.

Backflow generally occurs at high incidence [6], defined in Eqn. 3.1, where $\beta_\text{b}$ is the inlet blade angle measured from the tangential direction and $\beta$ is the inlet flow angle measured from the same direction. For axial flows, $\beta$ is the arctangent of the flow coefficient in a bulk flow analysis.

$$i = \beta_\text{b} - \beta$$

(3.1)

Significant backflow at the inducer leading edge induces unstable pump conditions [14,15], which affects the incidence angles and overall pump performance [6]. Pump performance is often measured with the head coefficient ($\psi$), defined in Eqn. 3.2, where $P_{00}$ is the inlet total pressure, $P_{01}$ is the outlet total pressure, $\rho$ is the fluid density, and $U_{\text{tip}}$ is the inducer tip speed $(\omega D_{\text{Tip}}/2)$. Changing incidence angles affect the rate of cavitation production and growth [80], which can lead to cavitation instabilities such as rotating cavitation or cavitation surge. Backflow that penetrates far upstream of the inducer restricts the upstream flow area, accelerating the inlet flow. This can lead to upstream cavitation, which grows and collapses with time [16]. Cavitation can further exasperate flow instabilities and reduce performance due to vapor volume growth on the inducer blades. The growing vapor volumes reduce effective flow area in the blade throat which alters flow diffusion as fluid enters the blade passages and changes how the blades are loaded. Cavitation instabilities can also impart unsteady forces to the inducer, potentially resulting in structural failure of the pump. Cavitation in inducers is often described using the cavitation number ($\sigma$), defined in Eqn. 3.3 where $P_v$ is the fluid vapor pressure. Cavitation formation is strongly dependent on the flow coefficient ($\phi$), defined in Eq. 3.4, where $m_{\text{in}}$ is the mass flow at the inducer inlet and $A_{\text{LE}}$ is
the frontal area of the inducer measured at the leading edge. Typically larger cavity volumes form at lower flow coefficients.

\[
\psi = \frac{P_{01} - P_{00}}{\rho U_{tip}^2} \tag{3.2}
\]

\[
\sigma = \frac{P_{00} - P_v}{\frac{1}{2} \rho U_{tip}^2} \tag{3.3}
\]

\[
\phi = \frac{\dot{m}_{in}}{(\rho A_{LE} U_{tip})} \tag{3.4}
\]

Japikse’s two-elements-in-series (TEIS) model is a useful way to understand inducer inlet flow [17]. This is a 1D flow model where the inducer inlet from upstream to the throat is considered as a diffuser. The area ratio (AR) of this diffuser is the ratio of the blade throat area to the upstream inlet relative flow area at the tip section and is found using Eqn. 3.5 where \( A_{throat} \) is the minimum area in a blade passage, and \( A_{inlet} \) is the bundle of streamtubes in the relative reference frame upstream of the blade leading edge that must contract to pass through the throat [17, 18].
\[ AR = \frac{A_{\text{throat}}}{A_{\text{inlet}}} = \frac{\sin \beta_b}{\sin \beta} = \frac{\sin \beta_b}{\sin (\arctan \phi)} \] (3.5)

The inlet area of the TEIS diffuser is proportional to \( \sin(\beta) \) while the discharge area is proportional to \( \sin(\beta_b) \) [17]. The inlet flow angle is equal to \( \tan^{-1}(\phi) \). The inlet blade angle is fixed at the blade tip. If the area ratio of the inlet diffuser in this TEIS system is too large then the inlet will stall [19]. Japikse showed that at a critical area ratio, \( AR_{\text{crit}} \), above \( \sim 1.5 \) the inlet will start to stall, yielding backflow [19, 20]. The ideal pressure recovery of this inlet diffusing section can be expressed as \( 1 - 1/AR^2 \); hence, as with all diffusers, higher area ratio means theoretically higher pressure recovery or higher static pressure. Inducers often operate close to this stall limit, because the high pressure recovery minimizes the growth of the vapor cavity and delays head breakdown [4].

Prior studies, both experimental and numerical, have explored the cause and behavior of backflow. Experimental work has shown that inlet reverse flow can occur near the casing wall due to a significant axial pressure gradient between the blades. This gradient causes the pressure rise to be produced in the first half of the inducer while leaving the latter half largely unloaded [21]. An axial momentum defect near the casing wall at the impeller inlet caused by the boundary layer also contributes to backflow formation [21], and is affected by tip clearance flow [22].

Computational studies have also explored the effect of tip clearance on backflow. Kim observed that backflow penetrates further upstream and performance declines rapidly for larger tip clearances, but offered no explanation of the cause [23]. Kim also found that for small tip clearances, backflow upstream propagation is reduced, but flow begins to separate at the hub in the middle of the passage, also reducing performance [23]. You et al. showed that larger tip clearance creates a larger tip vortex and creates negative pressure regions along the tip vortex in a linear cascade [24].

These studies attribute backflow to tip leakage flow, driven by the pressure gradient across the blade tip. Figure 3.1, adapted from [6] illustrates how the pressure gradient between the pressure and suction sides of the blade causes flow to reverse direction and move back over the blade between the blade tips and pump casing. This flow generates a vortex structure near the leading edge of the blade [6] called the tip vortex. At flow coefficients below design, the blades are more...
heavily loaded, causing a more pronounced tip leakage flow [23]. The objective of this work is to determine if backflow formation is driven by tip leakage flow or by some other means. In this work, backflow formation is examined by comparing the characteristics of backflow for both an inducer with tip clearance (TC) that allows tip leakage flow, and an inducer with no tip clearance (NTC) to prevent tip leakage flow. Thus, the effects of incidence and diffusion on backflow are assessed independent of tip leakage flow.

3.4 Methods

A four blade inducer with a tip blade angle of 7° and a design flow coefficient of \( \phi = 0.07 \) was analyzed in numerical simulations over a range of flow coefficients (\( \phi = 0.028, 0.042, 0.056, 0.07, 0.075, 0.08 \)). The base geometry with tip clearance (TC) has a tip clearance ratio (defined in Eq. 3.6) of \( \tau = 0.32\% \). The second inducer considered in this study with no tip clearance (NTC) is identical to the base geometry, however, the shroud radial position is reduced to be the same as the blade tip, resulting in zero tip clearance. Removal of tip clearance prevents tip leakage flow. A mid-plane section of the TC inducer mesh is shown in Fig. 3.2. Figures 3.3a and 3.3b are detail images of the tip meshes to clarify the difference between the considered inducer geometries. The commercial CFD package Star-CCM+ was used to test each inducer geometry over a range of flow coefficients and cavitation numbers. Flow coefficient and cavitation number were controlled using the inlet mass flow rate boundary condition and static pressure outlet boundary condition, respectively. The steady Reynolds-averaged Navies-Stokes equations were solved with a 2nd-order upwind convection scheme and segregated solver. The realizable k-\( \varepsilon \) model was used to model turbulence. This model applies wall functions to model the boundary layer in the viscous sublayer if the wall \( y^+ > 30 \), and assumes the mesh properly resolves the viscous sublayer for regions where the wall \( y^+ < 30 \).

\[
\tau = \frac{\text{Radial Tip Clearance}}{D_{tip}}
\]  

(3.6)

Lundgreen modeled the flow physics of the TC 7° inducer geometry with the same turbulence, cavitation, and physics models with good agreement when compared to experimental data, predicting the cavitation number where head breakdown occurs within 1% of the experimentally
observed value [16]. For this work, both single phase and cavitating simulations in water are considered. To model cavitation, an Eulerian multiphase volume of fluid homogeneous seed-based mass transfer model approach [81] was used. This method uses a simplification of the general Rayleigh-Plesset equation and solves the single phase governing equation set for an equivalent fluid with physical properties defined as functions of constituent phases and volume fractions. Individual cavitation bubbles are not modeled with this approach. A Rayleigh-Plesset formulation that includes the influence of bubble growth acceleration along with viscous and surface tension effects is employed to model rate of vapor production. For these unsteady, multiphase simulations, a time step of $1.388 \times 10^{-5}$ s was used with 5 inner iterations per time step with first order temporal discretization in an implicit unsteady model. This corresponds to 0.5° of rotation per time step which has been found in similar simulations to be sufficiently small for good simulation convergence [16]. While the maximum Courant number of any simulation for this work is located at a single cell at the blade tip and is quite high at 205, the typical Courant number is a much more reasonable 0.16, further suggesting the time step and mesh densities are acceptable.

While good simulation convergence of macro quantities has been shown with this modeling approach for similar inducer geometries [16], it should be noted that the realizable k-ε model used here to model turbulence may not be ideal for swirling flow characteristic of an inducer system and local phenomena presented here cannot necessarily be validated. Linear k-ε models are based on
the Boussinesq approximation, which implies a linear constitutive relation and does not account for rotation and streamline curvature effects on turbulence, assumes eddy-viscosity to be isotropic, and struggles to model non-equilibrium turbulent flow [82].

The inducer computational domains were defined by meshes generated in Star-CCM+. TC inducer meshes were generated previously during Lundgreen’s work [16], with NTC inducer meshes generated specifically for this work. To generate the NTC mesh, the shroud radial position was reduced to the same position as the blade tips. Figures 3.3a and 3.3b show how the radial
position of the shroud was reduced to remove the tip clearance and provides a comparison between the TC and NTC meshes. The meshes have $6.0 \times 10^6$ and $5.7 \times 10^6$ polyhedral cells for the TC and NTC inducers, respectively, with refinement near the inducer blades and hub, as seen in Fig. 4.2, to better capture the tip vortex. The TC mesh uses 8 prism layer cells and $\sim 2$ polyhedral cells to span the tip gap, and the average $y^+$ value for the meshes is 8. Lundgreen performed grid independence studies using the TC inducer geometry meshes and found that pump performance monitors, such as inducer head coefficient ($\psi$), varied by less than 0.35% when the mesh size was increased to $12.2 \times 10^6$ cells from $6.5 \times 10^6$ [16]. To verify Lundgreen’s grid independence study, a Richardson based error estimation using a safety factor of 1.25 [75, 76] was performed to determine the grid convergence index (GCI) using additional meshes of $12.0 \times 10^6$ and $23.5 \times 10^6$ cells for the TC inducer geometry and $11.9 \times 10^6$ and $25.3 \times 10^6$ cells for the NTC inducer geometry. The resulting GCIs are included in Table 3.1 and provide an error band estimation. GCI$_{12}$ shows the estimated error of the mid level mesh refinement when compared to the most refined mesh. GCI$_{23}$ shows the estimated error of the base mesh when compared to the mid level mesh. The difference in GCI between TC and NTC geometries stems from mesh differences in the refinement near the inducer blades and hub, partially visible in Figs. 3.3a and 3.3b, as the TC meshes were generated previously and used in other work. However, the error estimations using GCI are reasonably small for both geometries and indicate the baseline meshes are sufficiently refined and more refined TC meshes are not required. Thus, for this work, all reported values are estimations with an error band of 2.8% for the TC inducer and 0.28% for the NTC inducer.

Simulation convergence was determined case by case by evaluating the solution monitors of inlet mass flow, outlet mass flow, inlet total pressure, head coefficient, and rotordynamic forces on the inducer blades. NTC inducer simulations at the lowest flow coefficients simulated ($\phi = 0.028, 0.042$) produced quasi-steady solutions. In these cases, the solutions were considered

Table 3.1: Grid convergence index comparison for varying mesh sizes. These values loosely represent error bands for data extracted from the numerical simulations.

<table>
<thead>
<tr>
<th></th>
<th>TC Inducer</th>
<th>NTC Inducer</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCI$_{12}$ (%)</td>
<td>2.12</td>
<td>0.09</td>
</tr>
<tr>
<td>GCI$_{23}$ (%)</td>
<td>2.83</td>
<td>0.29</td>
</tr>
</tbody>
</table>
converged when the solution monitors exhibited the same periodic behavior over 10 blade revolutions. The steady simulation monitors at $\phi = 0.028$ and $\phi = 0.042$ were found to deviate from the unsteady values by less than 1%, further validating both single phase and cavitating simulation scenarios.

### 3.5 Results

In this section, the examination of backflow characteristics such as upstream penetration distance, mass flow at the inducer leading edge, and velocity profiles at various axial locations upstream of the inducer leading edge will serve to isolate backflow from tip leakage flow and will show that inlet diffusion is the primary mechanism for backflow, not tip leakage flow. Figure 3.4 shows the backflow upstream penetration distance normalized by the inducer blade tip diameter over the range of simulated flow coefficients for both the NTC and TC inducers. Upstream penetration distance is measured from the inducer blade leading edge to the furthest upstream axial location where fluid exhibits a velocity component opposite the core flow and is normalized with the inducer tip diameter. Both $L_p/D_{tip}$ curves follow the same general trend of increasing upstream backflow penetration with decreasing flow coefficient. However, the NTC inducer exhibits up to $0.5L_p/D_{tip}$ greater backflow penetration at all flow coefficients below $\phi = 0.056$. At $\phi > 0.07$, the TC inducer exhibits marginally higher backflow penetration than the NTC inducer, which is attributed to the tip clearance leakage flow.

Figure 3.5 shows how the normalized total amount of mass flow in the backflow region, $m_{\text{back}}/m_{\text{in}}$, at the leading edge varies with $\phi$. The backflow mass flow rate is the mass flow in the fluid region with a velocity component opposite that of the core flow. Here $m_{\text{back}}$ is computed on a plane at the inducer leading edge that encompasses the full span of the inducer blades. Both inducer geometries exhibit similar backflow mass flow rates at the leading edge below $\phi = 0.042$, with the TC inducer exhibiting more than 30% greater mass flow above $\phi = 0.042$. At high flow coefficients, the TC inducer exhibits a slightly larger backflow mass flow rate at the leading edge. This is due to the leakage flow contribution to the tip vortex. However, as the flow coefficient drops, the inlet flow diffusion gets larger, increasing the intensity of the stall, which dominates backflow formation, and marginalizes the effect of tip leakage flow causing both inducers to exhibit similar backflow mass flows.
Figure 3.4: Non-dimensional backflow penetration distance as a function of flow coefficient. The solid line is a fit to the NTC data while the dashed line is a fit to the TC data. At all $\phi < 0.7$ the NTC inducer experiences greater backflow upstream penetration. At $\phi > 0.7$ the TC inducer exhibits slightly greater backflow upstream penetration, although the backflow is vanishing for both inducers.

Examination of normalized axial velocity profiles at multiple upstream axial locations helps determine differences in the backflow between the two inducer geometries. Figures 3.6a, 3.6b, and 3.6c show normalized axial velocity profiles, $C_{\text{axial}}/U_{\text{tip}}$, for $\phi = 0.028$ and $\phi = 0.08$ at three normalized positions upstream of the leading edge, $z/D_{\text{tip}} = 0.5$, 1.5, and 3. The profiles are shown as a function of percent span from hub to tip. $\phi = 0.028$ and $\phi = 0.08$ are the the small and large limits of simulated off-design flow coefficients, respectively, and show the greatest variation in backflow penetration. At the higher flow coefficient, $\phi = 0.08$, the curves are nearly the same for both geometries and at all $z/D_{\text{tip}}$ locations. However, at $\phi = 0.028$, significant upstream backflow penetration is present and notable variation exists in the shape of the velocity profile, velocity magnitude, and the span location where backflow begins to occur between the NTC and TC geometries. At all considered axial locations the largest magnitude of backflow velocity near the casing wall is greater (47.8%, 69.5%, and 43.1% greater at the $z/D_{\text{tip}} = 0.5$, 1.5, and 3 locations respectively) for the NTC inducer.
Analysis of the span location where flow reverses direction helps describe the backflow profile; namely how the radial thickness of the backflow region varies with upstream position. Near the leading edge of the blades \( (z/D_{tip} = 0.5) \) and at \( \phi = 0.028 \) for the NTC inducer, the radial thickness of backflow extends nominally from 85\% to 100\% span. At \( z/D_{tip} = 1.5 \) the backflow region extends nominally from 80\% to 100\% span and it increases at the \( z/D_{tip} = 3 \), extending from 70\% to 100\% span. In contrast, for the TC inducer \( (\phi = 0.028) \) the radial thickness of the backflow region extends from nominally 80\% to 100\% span at \( z/D_{tip} = 0.5 \), increasing to a range of 75\% to 100\% span at both the \( z/D_{tip} = 1.5 \) and 3 locations. Typical backflow is shown in Figs. 3.7a and 3.7b, which depict both inducer geometries operating at \( \phi = 0.042 \); the backflow is shown in dark blue, and depicts the full recirculation region as flow propagates upstream from the inducer leading edge as backflow until it again becomes entrained in the core flow. It is clear backflow extends much further upstream in the NTC inducer for this flow coefficient. While Figs. 3.7a and 3.7b depict a single flow coefficient, the shape of the backflow is consistent for both inducer geometries over all considered flow coefficients. For both inducer geometries, backflow thickness increases with increasing upstream position up to a maximum, beyond which backflow thickness decreases. However, above \( z/D_{tip} > 1.5 \), the backflow region for the NTC inducer scenario always comprises a larger percentage of the passage span than for the TC inducer.
Figure 3.6: Nondimensional axial velocity profiles as a function of percent span (hub to shroud) for axial upstream locations: (a) $z/D_{tip} = 0.5$, (b) $z/D_{tip} = 1.5$, and (c) $z/D_{tip} = 3$
Figure 3.7: Contour plots of axial velocity magnitude in a cross sectional view of both inducer geometries operating at $\phi = 0.042$. Axial velocity is colored with a red-blue color bar. Backflow extends further upstream in the NTC case: (a) NTC inducer and (b) TC inducer.
The inlet tip vortex is now considered. The inlet tip vortex is distinct from, and smaller than the backflow and is often confused with backflow. Figures 3.8a and 3.8b plot streamlines upstream of the leading edge of the inducer colored by $\lambda_2$ criterion for both geometries. This helps to clarify the difference between the tip and backflow vortices. $\lambda_2$ is defined as the second eigenvalue of the strain-rate and spin tensors, and values of $\lambda_2 < 0$ can be interpreted as vortex regions. This approach has been shown to be successful in identifying vortex formation in a large variety of flows [83]. The NTC inducer shown in Fig. 3.8a has been truncated at $r/R = 0.995$, where $r$ is the radial position and $R$ is the shroud radius, to allow for better visualization of the tip vortex. The TC inducer shown in Fig. 3.8b is unaltered. The tip vortex is clearly seen for both inducers as the bundle of streamlines with large negative $\lambda_2$ values just upstream of the blade near the blade tip. While not as closely bundled as the tip vortex streamlines, parts of the backflow vortex can be seen slightly upstream of the tip vortex for both geometries and is defined by another region of large negative $\lambda_2$ values. The core of the backflow vortex is not shown in either Fig. 3.8a or 3.8b, as it is located further upstream than what is shown in Fig. 3.8a and 3.8b. The tip vortex remains near the blade leading edge, but the backflow extends far upstream, and has an entirely different shape. The tip vortex is created from the interaction of two orthogonal or quasi-orthogonal velocity gradients, one caused by the casing boundary layer, and the other due to the reduction in velocity as the flow approaches the blade leading edge. The tip leakage flow present in the TC inducer further feeds the tip vortex, causing it to extend further upstream than that produced by the NTC inducer, a difference that is clear when comparing Figs. 3.8a and 3.8b. The size difference of the tip vortex between the two geometries is shown in Fig. 3.9, which plots normalized tip vortex volume as a function of flow coefficient. The tip vortex volume is defined by any cell for which $\lambda_2 < 0$ in a region starting at the inducer leading edge and extending 17 mm upstream of the inducer leading edge. At 17 mm upstream the tip vortex is completely captured for all cases. Here, the tip vortex volume is normalized by the cube of the tip diameter. At all flow coefficients simulated, the tip vortex of the TC inducer is larger than that of the NTC inducer. Below design ($\phi < 0.07$), the TC inducer exhibits a tip vortex with 28% greater volume on average, than that produced by the NTC inducer. Above the design flow coefficient, the size difference increases drastically due to the small magnitudes involved.
Figure 3.8: Streamlines plotted just upstream of the leading edge of the TC inducer operating at $\phi = 0.042$. $\lambda_2$ criterion is colored with a red-blue color bar. Regions where $\lambda_2 < 0$ can be interpreted as vortex regions. Regions where $\lambda_2 \geq 0$ have no physical interpretation. This plot differentiates the tip vortex from the backflow. The NTC inducer has been truncated at $r/R = 0.995$ to allow for better visualization of the tip vortex: (a) NTC inducer and (b) TC inducer.
Figure 3.9: Normalized tip vortex volume as a function of flow coefficient for both inducers. At all flow coefficients, the NTC tip vortex is larger than the TC tip vortex.

The strength of the tip vortex is quantified by taking the surface integral of tangential vorticity over the surface of the tip vortex volume to determine circulation. Figure 3.10 shows the circulation ($\Gamma$) of the tip vortex normalized by tip diameter and speed as a function of $\phi$ for both the NTC and TC inducers. For all flow coefficients, the tip vortex of the TC inducer has much greater circulation than the NTC inducer. At $\phi = 0.028$ the TC inducer exhibits a 45% greater circulation than the NTC inducer. Circulation for the NTC inducer increases with increasing $\phi$ until $\phi = 0.056$, at which point the tip vortex circulation is 124% greater than the NTC tip vortex. Above $\phi = 0.056$, the circulation for the NTC inducer decreases rapidly until $\phi = 0.08$. However, due to the relatively low magnitude of the NTC tip vortex circulation, the TC tip vortex circulation is 186% greater than the NTC tip vortex circulation at $\phi = 0.08$. The NTC tip vortex circulation does not follow the same trend as the TC tip vortex, but instead decreases gradually with increasing $\phi$. The TC inducer generates a stronger tip vortex than the NTC inducer at all flow coefficients because of the influence of the tip leakage flow, which proceeds from the pressure side to suction side of the blade in the upstream direction. This mechanism imparts additional energy to the tip vortex, increasing the size and strength.

As flow coefficient is reduced, $\beta$ decreases, and incidence increases. Increases in incidence cause the flow to accelerate more around the leading edge, and in conjunction with the diffusion
Figure 3.10: Normalized tip vortex circulation as a function of flow coefficient for both inducers explored. At all flow coefficients, the TC tip vortex has greater circulation than the NTC tip vortex.

Figure 3.11: Pressure recovery plotted as a function of flow coefficient. The NTC inducer experiences a greater pressure recovery, at all considered flow coefficients.
required to equalize streamline velocities, the flow separates from the blade surface and induces stall and flow recirculation [17, 20, 84]. The point at which the inducer blades stall and generate backflow corresponds to a specific pressure recovery. Therefore, the maximum pressure recovery \((C_{p,\text{max}})\) produced by an inducer can be used as a rough measure of the propensity of the inducer to stall. \(C_{p,\text{max}}\) is defined in Eqn. 3.7, where \(P\) is the maximum static pressure on the shroud wall produced by each inducer over the range of flow coefficients considered. The location of \(P\) is on average 0.4 tip diameters downstream of the leading edge for the NTC inducer and varies by up to 67\% over the range of flow coefficients. The location of \(P\) is on average 0.33 tip diameters downstream of the leading edge for the TC inducer and varies by up to 14\% over the range of flow coefficients. Because both considered inducer geometries have the same blade angle, incidence increases are equal between the considered geometries, making \(C_{p,\text{max}}\) a direct comparison of propensity to stall between the two inducers. Figure 3.11 shows \(C_{p,\text{max}}\) as a function of \(\phi\) for both inducers. \(C_{p,\text{max}}\) increases with decreasing flow coefficient to 0.57 for both inducer geometries, but for all flow coefficients the NTC inducer exhibits up to 15\% greater pressure recovery locally. The pressure recovery is lower for the TC case because the tip clearance unloads the blade tip and reduces the achievable pressure rise. The NTC inducer, due to the higher pressure recovery, has a greater propensity to stall and generate backflow, which is corroborated by Fig. 3.4. Decreasing inlet flow angle also causes an increase in \(AR\). Therefore, larger \(C_{p,\text{max}}\) values correspond to larger \(AR\), and higher inlet diffusion. When comparing Figs. 3.11 and 3.4, high backflow penetration is produced at the same flow coefficients that high \(C_{p,\text{max}}\) are seen. That is, both backflow penetration and \(C_{p,\text{max}}\) increase with decreasing flow coefficient. The flow phenomenon resulting in a high \(C_{p,\text{max}}\) is inlet diffusion causing the flow to separate, which points to inlet diffusion as the root cause of backflow formation.

The greater pressure recovery of the NTC inducer results in greater pump performance as seen in Fig. 4.3, which provides dimensionless performance curves for both inducer geometries. At all simulated flow coefficients, the NTC inducer produces a higher head, ranging from a 4\% increase at \(\phi = 0.028\) to a 19\% increase at \(\phi = 0.08\). The pump performance difference between the two geometries is again explained by the tip clearance unloading the blade tip and reducing achievable pressure rise in the TC case.
Figure 3.12: Non-dimensional performance curves for both inducer geometries. The solid line is a fit to the NTC data while the dashed line is a fit to the TC data. At all $\phi$ the NTC inducer experiences greater head production than the TC inducer.

$$C_{p,max} = \frac{P - P_{inlet}}{1/2 \rho U_{tip}^2}$$  \hspace{1cm} (3.7)

Analysis using $AR$ is perhaps more descriptive than analysis using $\phi$ when considering flow diffusion at the inducer blade leading edge. $AR$ provides a simplified description of flow diffusion at the inducer inlet, and accounts for inducer inlet blade angle and the flow coefficient in its definition (Eq. 3.5) to describe incidence. As incidence increases in an inducer, the flow is required to accelerate more around the leading edge, which results in a strong diffusion to bring the highly accelerated surface streamline into balance with the adjacent streamlines [84]. The strong diffusion causes flow to separate from the blade surface, and can cause it to recirculate and propagate upstream as backflow. In other words, as the boundary layer on the inducer blades thickens with increasing incidence, the adverse pressure gradient can cause the flow to separate from the wall and propagate upstream [85]. For the flow to separate in this manner, the momentum in the streamlines adjacent to the wall is insufficient to overcome the adverse pressure gradient and viscous shear stresses along the wall [84]. Figure 3.13 shows an example case of separated flow propagating upstream of the inducer leading edge as backflow due to the pressure distribution of the inducer [17].
Figure 3.13: Typical backflow formation in a centrifugal impeller. The pressure distribution of the impeller has forced the separated flow upstream of the leading edge of the blades. From [86].

Additionally, the streamlines approaching the region of separated flow are deflected from the blade surface, potentially increasing the size of the backflow. Backflow upstream propagation is precipitated by separated flow caused by flow diffusion near the inducer blade leading edge.

Analysis using AR also provides a connection to the TEIS model through its 1-D characterization of inlet diffusion. Figures 3.14 and 3.15 show the relationship between AR and backflow penetration and mass flow, respectively. Figure 3.14 plots the normalized backflow penetration distance as a function of AR. A direct relationship between backflow penetration distance and AR is evident. The magnitude of the backflow penetration distance is $L_p/D_{tip} \approx 0.1$ at $AR = 1.5$ for
both NTC and TC cases, but then increases to maxima of $L_p/D_{tip} \approx 5.4$ for the NTC case and $L_p/D_{tip} \approx 4.6$ for the TC inducer at $AR = 4.35$. Again, the NTC case exhibits up to 14% greater backflow penetration. Recall that Japikse posited a critical area ratio where backflow would occur at a value of $AR_{crit} = 1.5$ based on test data. This predicted limit closely matches the CFD results shown in Fig. 3.14. Very low backflow is apparent at $AR \approx 1.5$, but backflow penetration increases steadily above this critical value. The strong relationship between backflow penetration and $AR$ further suggests inlet diffusion is the main force behind backflow formation.

Figure 3.15 provides the normalized backflow mass flow at the leading edge as a function of $AR$. The backflow mass flow at $AR \approx 1.5$ is minimal, but increases linearly to 0.6 with increasing $AR$ for both inducers. At all considered $AR$, the TC inducer exhibits a greater backflow mass flow than the NTC inducer. However, little difference between the two inducers is present at high $AR$ but the difference increases with decreasing $AR$ to a maximum of 87% at $AR = 1.53$. This is due to the additional mass flow added by the tip leakage flow in the TC geometry which contributes to the tip vortex, as previously discussed. While the backflow mass flow is greater in the TC case, when Figs. 3.14 and 3.15 are compared, it is clear the additional backflow mass flow at the leading edge does nothing to contribute to the backflow upstream propagation. This further suggests that the backflow structure and tip vortex are separate, distinct phenomenon. The strong relationship between backflow and $AR$ seen in Figures 3.14 and 3.15 emphasizes the importance of diffusion in backflow development. As $AR$ increases, the flow diffuses until a critical value is reached and the inducer stalls, which is the primary mechanism for backflow [4]. Again, features of backflow exhibit a strong dependence on $AR$, which describes inlet diffusion. It is clear that inlet diffusion plays a major role in backflow formation and development.

Under cavitating conditions the throat area of an inducer is reduced by cavity formation, which decreases $AR$ with decreasing $\sigma$ for a constant $\phi$. The cavitating area ratio ($AR_c$) accounts for the reduced throat area, and is defined here by multiplying the single phase area ratio by the corrective term $1 - T_c$, where $T_c$ is the area average of the volume fraction of vapor over the throat area. The corrective term $1 - T_c$ therefore describes how much the throat area is reduced relative to the unblocked throat area and provides a percentage reduction for the single phase $AR$ for the inducer operating at a given cavitation number. Cavitation blockage in the throat is defined in Eqn. 3.8.
Figure 3.14: Non-dimensional backflow penetration as a function of $AR$ for all simulated flow coefficients. Polynomial curve fits for each data set are shown. The curve fit for the NTC data has a $R^2 = 0.9998$, while the TC data fit has an $R^2 = 0.9999$.

Figure 3.15: Non-dimensional backflow mass flow as a function of $AR$ for all simulated flow coefficients. Linear curve fits for each data set are shown. The curve fit for the NTC data has a $R^2 = 0.9982$, while the TC data fit has an $R^2 = 0.9981$. 
Figure 3.16: Non-dimensional backflow penetration as a function of $AR_c$. A solid line is fit through all the data, for which $R^2 = 0.9431$.

$$T_c = \frac{\int \phi_{vapor} dA}{A_{throat}} \quad (3.8)$$

At high $\sigma$ when cavity volumes are small, $AR_c$ approaches $AR$ as there is no reduction in throat area due to blockage, resulting in the maximum possible throat area (Eqn. 4.5). At low $\sigma$, significant cavitation blockage forms, reducing the throat area, and producing an $AR_c$ that can be significantly smaller than $AR$. The reduced $AR_c$ at low $\sigma$ corresponds to comparatively low inlet diffusion as the inlet area remains constant while the throat area is reduced. Figure 3.16 shows how backflow penetration varies with $AR_c$ for all simulated flow coefficients and cavitation numbers for both TC and NTC inducer geometries. Backflow penetration and $AR_c$ values were time-averaged over 5 inducer revolutions. Multiple $\sigma$ were simulated at each flow coefficient, providing a large range of cavity volumes, thereby creating a wide spread of $AR_c$ values.

The majority of points falling between $1.3 \leq AR_c \leq 1.75$ in Fig. 3.16 exhibit less than 0.5 tip diameters of backflow upstream penetration. These points consist mainly of data from simulations run at $AR = 1.53$ and $AR = 1.75$. At $AR = 1.53$, the inducer is operating above the design point, which generates minimal cavitation and throat blockage. The low throat blockage causes a difference of 15.1% between max and min $AR_c$, which is why all the points corresponding
to $AR = 1.53$ are clustered around $AR_c \approx 1.4$. If $AR_c$ is near its maximum for a given $AR$, inlet diffusion remains relatively high for that $AR$. Although inlet diffusion at $AR = 1.53$ is relatively high for the $AR$ and essentially constant over the range of simulated $\sigma$ as there is little to no reduction due to blockage, inlet diffusion for $AR = 1.53$ is low relative to other simulated $AR$, which results in minimal backflow relative to other $AR$ and is apparent in Fig. 3.16. Figure 3.17 provides another perspective of how $AR_c$ varies with $\sigma$ for all simulated $AR$. The $y$ axis of Fig. 3.17 is the same as the $x$ axis of Fig. 3.16. As $\sigma$ decreases for $AR = 1.53$, the 15.1% variation in maximum and minimum $AR_c$ values can be seen, suggesting that at this $AR$, cavity volumes remain relatively constant in the throat despite significant drops in inlet total pressure that cause $\sigma$ to fall (Eq. 3.3).

Another clustering of points at low $AR_c$ is seen in Fig. 3.16 for simulation data at $AR = 1.75$. This $AR$ is the design point, and the inducer exhibits accordingly small throat blockage and backflow, although there is slightly more cavitation blockage than seen at $AR = 1.53$. This throat blockage causes a difference between maximum and minimum $AR_c$ of 24.8%. The relatively low variation of $AR_c$ again causes the clustering of points at low $AR_c$. Each point in this cluster exhibits backflow penetration less than 0.6 tip diameters upstream. This is again a function of low blockage resulting in relatively high inlet diffusion for $AR = 1.75$ as $AR_c$ approaches its maximum. As with $AR = 1.53$, inlet diffusion for $AR = 1.75$ is mainly limited by the low $AR$. Figure 3.17 explicitly shows the 24.8% variation in $AR_c$ as $\sigma$ decreases for $AR = 1.75$. This trend is similar to that seen for $AR = 1.53$, but due to the minor increase in throat blockage, $AR_c$ is varies slightly more with $\sigma$ for $AR = 1.75$ than for $AR = 1.53$. While cavitation formation is relatively constant with $\sigma$ for $AR = 1.53$, a trend of exponential cavitation formation below a specific $\sigma$ begins to become evident at $AR = 1.75$.

As $AR$ increases to 2.9 (60% of design) the cavitation likelihood increases because of backflow blockage. This causes greater throat blockage at low $\sigma$, resulting in a difference of 53.8% between maximum (2.9) and minimum (1.34) $AR_c$. The reduction in $AR_c$ from the maximum causes a decrease in backflow penetration from 4.2 tip diameters at $AR_c = 2.9$ to 0.03 tip diameters at $AR_c = 1.34$. The large disparity between upstream backflow penetration at the maximum and minimum $AR_c$ for simulations at $AR = 2.9$ highlights the strong dependence of backflow penetration on inlet diffusion, as $AR_c$ is determined by throat blockage, which dictates the inlet flow
diffusion. Despite the large difference between maximum and minimum $AR_c$, as seen in Fig. 3.17 many of the points simulated at $AR = 2.9$ remain around $AR_c \approx 2.9$.

The tendency of these points to remain near the maximum $AR_c$ is again seen Fig. 3.17. For $AR = 2.9$, $AR_c$ remains constant until $\sigma \approx 0.05$. Below this $\sigma$, $AR_c$ decreases. Therefore, for the considered inducers at $AR = 2.9$, cavity volumes do not increase linearly with decreasing $\sigma$, but rather remain somewhat constant until a critical $\sigma$ is reached, below which cavity volumes in the throat increase rapidly. Constant $AR_c$ values for $AR = 2.9$ above $\sigma = 0.05$ while backflow penetration varies is a function of the definition of $AR$ and $AR_c$. Both $AR$ and $AR_c$ assume uniform upstream flow conditions which are compared to flow conditions in the throat, but cavitation blockage does not occur solely in the throat. Figure 3.18 illustrates the difference in cavity formation for two points at the same flow coefficient that also exhibit the same $AR_c$. Figure 3.18a shows an iso-surface where $\phi_{vapor} = 0.1$ for the NTC inducer operating at $AR = 2.9$, $\sigma = 0.025$ and $AR_c = 2.9$. Significant cavity volumes are present upstream of the blade leading edge and the throat area. The same $AR_c = 2.9$ is reported for the inducer operating at $AR = 2.9$ and $\sigma = 0.157$, with an iso-surface of $\phi_{vapor} = 0.1$ shown in Fig. 3.18b, but cavity volumes upstream of the blade leading edge are much smaller than in Fig. 3.18a. Figure 3.18c shows an iso-surface of $\phi_{vapor} = 0.1$ for the TC inducer. Again, $AR$ and $AR_c = 2.9$, but here $\sigma = 0.062$. Cavity volumes upstream of the leading edge are relatively large at this operating point. As seen in Fig. 3.17, this is the lowest $\sigma$ for which the TC inducer operating at $AR = 2.9$ exhibits an $AR_c = 2.9$. The lowest $\sigma$ at which $AR_c = 2.9$ for $AR = 2.9$ is greater in the TC case than the NTC case, which means that instead of forming upstream as in the NTC inducer, cavitation forms in the throat of the TC inducer, which causes the $AR_c$ reduction at higher $\sigma$ evident in Fig. 3.17. Figure 3.18d shows the $\phi_{vapor} = 0.1$ iso-surface for the TC inducer operating at $AR = 2.9$, $AR_c = 2.9$, and $\sigma = 0.222$. Cavitation formation on the suction side of the blades is much smaller than in Fig. 3.18c, but because $AR_c$ is equal between the two cases, cavitation blockage in the throat is the same. Cavity volumes upstream of the throat can change upstream flow conditions and inlet diffusion while leaving $AR_c$ unaffected. This accounts for the vertical lines of data in Fig. 3.16 near $AR_c = 1.75, 2.9$ and 4.35. While using $AR$ and $AR_c$ as measures of inlet diffusion does not account for changes in upstream flow conditions, it is still useful to indicate inlet diffusion trends with large scale $\sigma$ changes.
Scatter in $AR_c$ values increases with increasing $AR$ as cavitation propensity increases with $AR$. At low $AR$, cavitation volumes are small, which causes minimal changes in $AR_c$. Increasing $AR$ increases cavitation volumes and variability in $AR_c$. The greatest difference between maximum and minimum $AR_c$ is apparent for $AR = 4.35$. As can be seen in Fig. 3.16, $AR_c$ ranges from a maximum of 4.35 to a minimum of 1.35, a difference of 69%, which is greater than any other simulated $AR$. At the maximum $AR_c$, the inducer exhibits a backflow penetration of 6.57 tip diameters, while the inducer only exhibits a backflow penetration of 0.03 at the minimum $AR_c$. The large backflow at high $AR_c$ forms because of the high inlet diffusion possible due to high $AR_c$. The drastic decrease in backflow penetration is again a function of the large $AR_c$ disparity, due to increasing throat blockage and reducing inlet diffusion. The change in $AR_c$, is again shown in Fig. 3.17. At $AR = 4.35$, cavity volumes are constant until $\sigma \approx 0.04$, below which cavity volumes increase drastically, increasing blockage and reducing $AR_c$. This suggests that cavity volumes begin to form at similar inlet pressures but continue to increase in volume at a faster rate with decreasing $\sigma$ for $AR = 4.35$ than for any other $AR$. The constant cavity volume in the throat above $\sigma \approx 0.04$ again results in a constant $AR_c$ while backflow changes which is a function of changing upstream flow conditions due to cavitation upstream of the throat.

When considering the data from all simulated $AR$ and geometries as a whole, the trend shown in Fig. 3.14 of backflow increasing with $AR$ is unmistakable. This relationship continues to hold reasonably well for cavitating simulations with $AR_c$ in Fig. 3.16. Both inducer geometries exhibit very little backflow near the critical $AR_c = 1.5$, but increase with increasing $AR_c$. Backflow penetration increases almost linearly up to a maximum of 6.57 tip diameters at $AR_c = 4.35$. The direct relationship between increasing $AR_c$ and increasing backflow holds for all considered variables. This strong dependence on $AR$, regardless of flow coefficient, inducer geometry or cavitation blockage, solidifies inlet diffusion, described by $AR$ or $AR_c$, as the source of backflow formation.

### 3.6 Conclusions

This work investigated the effects of flow coefficient, tip leakage flow, and area ratio on the inlet flow field of an inducer under single phase and cavitating conditions in an effort to determine the driving force in backflow formation. In cases where backflow is present two vortex structures upstream of the leading edge are evident: a small tip vortex and a larger backflow region. Re-
moving the inducer tip clearance to prevent tip leakage flow was shown to increase the backflow upstream penetration up to 14% when compared to the case with tip clearance and leakage flow. At all flow coefficients below the design point, the inducer that experienced tip leakage flow exhibited less backflow upstream penetration. This suggests that tip leakage flow has a negligible effect in backflow formation. The tip leakage flow instead contributes to the strength of the tip vortex that is produced by the interaction of the casing boundary layer with the leading edge of the inducer. This tip vortex is often confused with the larger backflow structure in literature. $AR$ is found to be a useful metric in understanding backflow formation and describes how the flow enters the inducer. Large $AR$ values correspond to high inlet diffusion, and small $AR$ values illustrate low inlet diffusion. When an inducer is subject to cavitation, $AR_c$ is used to describe the reduced $AR$ that occurs when cavity formation reduces the throat area. Backflow upstream propagation is found to always increase with increasing $AR$ or $AR_c$ in both single phase and under cavitating conditions where the effective flow area in the throat is reduced. The strong relationship between backflow and $AR$ indicates the primary mechanism for backflow is high inducer inlet diffusion due to the high flow incidence at the leading edge. This correlation coupled with the fact that an inducer that allows
Figure 3.18: Isosurface views of $\phi_{\text{vapor}} = 0.1$ for both inducer geometries. Cavity volumes affect upstream flow conditions and introduce error into $AR_c$ calculations: (a) NTC inducer, $AR = 2.9$, $AR_c = 2.9$, $\sigma = 0.025$; (b) NTC inducer, $AR = 2.9$, $AR_c = 2.9$, $\sigma = 0.157$; (c) TC inducer, $AR = 2.9$, $AR_c = 2.9$, $\sigma = 0.062$; (d) TC inducer, $AR = 2.9$, $AR_c = 2.9$, $\sigma = 0.222$.

tip leakage flow does not increase backflow formation and growth over an inducer that prevents tip leakage flow provides a better basis of understanding of backflow formation in axial inducer systems.
CHAPTER 4. CRYOGENIC CAVITATION PERFORMANCE OF AN AXIAL INDUCER WITH A STABILITY CONTROL DEVICE

This chapter is a paper to be submitted to the Journal of Fluids Engineering of the American Society of Engineers. The formatting of this paper has been modified to meet the stylistic requirements of this dissertation.

4.1 Contributing Authors and Affiliations

Tate Fanning, Steven Gorrell, Daniel Maynes, Department of Mechanical Engineering, Brigham Young University, Provo, UT 84602, USA. Kerry Oliphant, Concepts NREC, White River Junction, Vermont 05001, USA.

4.2 Abstract

Performance improvements in turbopump systems pumping cold water have been obtained through implementation of a recirculation channel called a stability control device (SCD). This paper considers inducer performance at on and off design flow coefficients with and without an SCD where liquid hydrogen is the working fluid. Relevant thermodynamic effects present in liquid hydrogen at cryogenic temperatures (∼21 K) are modeled using CRUNCH CFD. The results reveal that the SCD yields marginal changes in the head coefficient. However, a stabilizing effect occurs at all considered flow coefficients, where a reduction in backflow occurs over much of the pump operational range. This occurs due to the SCD maintaining consistent, low incidence angles at the inducer leading edge. Thus, use of an SCD widens the operating range of the considered inducer, and allows for stable operation at flow coefficients far below the design point, even when liquid hydrogen is the working fluid.
4.3 Introduction

Inducers are often placed just upstream of a centrifugal pump to improve suction performance and reduce cavitation formation throughout the pump. While cavitating inducers are designed to operate with vapor formation on the suction surfaces of the blades, excessive cavity growth at flow coefficients below design can reduce pump head and impart undesirable rotordynamic forces to the inducer. At the cryogenic working temperatures of liquid hydrogen, fluid properties, most notably vapor pressure, vary significantly with small changes in temperature, and cavity formation introduces thermal effects. At these temperatures, the ratio of liquid to vapor density is low which requires a greater mass of liquid to vaporize to form a cavity. This emphasizes the effects of evaporative cooling, which reduces fluid temperature surrounding the bubble [25]. As vapor pressure is highly dependent on temperature in a cryogenic fluid, reduced vapor pressure in the fluid surrounding the cavity accompanies the drop in fluid temperature. The reduced vapor pressure effectively suppresses cavitation and improves pump performance. This is known as thermodynamic suppression head (TSH) [6, 26, 27]. The effects of TSH on NPSH are defined using Eq. 4.1 where NPSH is the net positive suction head, defined in Eq. 4.2. NPSH is a measure of excess fluid total pressure above fluid vapor pressure, and effectively describes the propensity of the working fluid to cavitate, with lower NPSH values indicating a higher likelihood of cavitation. A lower \( NPSH_{\text{required}} \) is desirable as the pump can then operate at a lower NPSH while still avoiding cavitation.

\[
(NPSH)_{\text{required}} = (NPSH)_{\text{ideal}} - TSH
\]  
(4.1)

\[
NPSH = \frac{P_t - P_{\text{vapor}}}{\rho g}
\]  
(4.2)

Due to the complexity and cost of working with cryogenic fluids, much of the work done to understand the effects of cavitation on the performance of inducer systems designed for cryogenic liquid turbopumps has been performed with cold water as the working fluid. Cold water does not exhibit the thermal effects caused by significant changes in fluid properties with marginal temperature changes, and does not provide an accurate representation of inducer performance under
real world cryogenic conditions. Understanding and accurately simulating the thermal effects of cavitation in cryogenic fluids is necessary for improved design of cryogenic liquid turbopumps.

Experimental studies exploring the thermal effects of cavitation and cavitation performance under cryogenic conditions have long been available [29–37], and attempts have been made to predict TSH effects for various fluids in pumps [37, 38]. The standard way to determine TSH for an inducer pumping liquid hydrogen is somewhat ambiguous and is based on experimental data of inducers operating around 21 K. It has been previously shown that inducers pumping liquid hydrogen can operate at a critical NPSH equal to an inlet-velocity head ratio of one, defined in Eq. 4.3, where \( Cm \) is the average meridional velocity [39]. This relationship is only true when the inlet-velocity head is small relative to the vapor head. Then, assuming the inducer operates at \( \phi < 0.15 \), a fluid with TSH can operate at an NPSH equal to an inlet-velocity head ratio of three for the same conditions. Using Eq. 4.1, the definition for TSH in liquid hydrogen is then defined by Eq. 4.4. This method is only a rough estimator of TSH that does not consider fluid property changes with temperature and is only valid when the inlet velocity head is small relative to the vapor head.

\[
NPSH = \frac{Cm^2}{2g} \tag{4.3}
\]

\[
TSH_{LH_2} = \frac{Cm^2}{g} \tag{4.4}
\]

It is only recently that efforts to simulate cryogenic cavitation using computational fluid dynamics (CFD) have been made [25, 40–44]. These efforts have resulted in a multi-element unstructured numerical framework called CRUNCH CFD [44]. This code solves the energy equation for a multiphase mixture in conjunction with conservation of mass and momentum, while considering the evaporative cooling effects of cavitation [40]. This approach has been shown to slightly under predict cavitation performance of an inducer with liquid hydrogen as the working fluid [41]. However, considering the complexity of the problem and the sensitivity of the inducer to small changes in flow rate, the agreement between results derived from CRUNCH CFD and experimental data is reasonably good, and provides a much improved estimation of the resulting flow field when compared to typical cold water CFD simulations.
Despite the improved cavitation performance at cryogenic temperatures due to thermal effects, additional cavitation instabilities develop when an inducer operates below the design flow conditions of the pump. These instabilities introduce unstable flow conditions and large rotordynamic forces, which can reduce performance and potentially cause pump failure [6]. Efforts have been made to combat these instabilities at off-design flow conditions. One such attempt is a recirculation channel called a stability control device (SCD) [87–89] that extracts high pressure fluid from a bleed slot just downstream of the inducer leading edge and reinjects the fluid into the upstream core flow. A longitudinal cross section of a flat plate inducer with an SCD is included in Figure 4.1 in which the bleed and reinjection slots are labeled. SCD implementation has been shown to delay cavitation formation in certain flow regimes, and improve inducer performance [16]. SCD utilization can also allow an inducer to operate far below the design flow condition by suppressing cavitation instabilities, which reduces rotordynamic forces and stabilizes the inducer [16, 53, 90]. These prior studies exploring SCD effectiveness have all used water as the working fluid, and have neglected the thermal effects of cavitation at cryogenic temperatures. Simulating inducer performance with an SCD using CRUNCH is a novel effort that offers a more accurate and realistic view of the effects of SCD implementation with an inducer.
Table 4.1: Geometric details of considered flat plate inducer.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip helix angle from axial direction (deg)</td>
<td>80.6</td>
</tr>
<tr>
<td>Rotor tip diameter (cm)</td>
<td>12.65</td>
</tr>
<tr>
<td>Rotor hub diameter (cm)</td>
<td>6.29</td>
</tr>
<tr>
<td>Hub tip ratio</td>
<td>0.496</td>
</tr>
<tr>
<td>Number of blades</td>
<td>3</td>
</tr>
<tr>
<td>Axial length (cm)</td>
<td>5.08</td>
</tr>
<tr>
<td>Peripheral extent of blades (deg)</td>
<td>280</td>
</tr>
<tr>
<td>Tip chord length (cm)</td>
<td>31.37</td>
</tr>
<tr>
<td>Hub chord length (cm)</td>
<td>16.15</td>
</tr>
<tr>
<td>Solidity at tip</td>
<td>2.350</td>
</tr>
<tr>
<td>Tip blade thickness (cm)</td>
<td>0.254</td>
</tr>
<tr>
<td>Hub blade thickness (cm)</td>
<td>0.483</td>
</tr>
<tr>
<td>Tip clearance (cm)</td>
<td>0.064</td>
</tr>
<tr>
<td>Ratio of tip clearance to blade height</td>
<td>0.020</td>
</tr>
</tbody>
</table>

The purpose of this work is to determine the effects of SCD utilization on an inducer with liquid hydrogen as the working fluid while considering the rapid changes in fluid properties with temperature that cause TSH, and the corresponding effects on inducer cavitation performance. The effects of SCD implementation are evaluated here by comparing the head coefficient, force coefficient, incidence, and backflow formation of a single inducer operating with and without an SCD.

4.4 Methods

A three bladed flat plate helical inducer with a tip helix angle of 80.6° as measured from the axis of rotation is the subject of numerical simulations here. The inducer tip diameter is 12.65 cm and has a hub to tip ratio of 0.5. Tip and hub diameters are constant along the rotor. Geometric features are shown in Table 4.1. This inducer was experimentally tested with no SCD by Meng and Moore [29], who provide cavitation performance results with liquid hydrogen. These experimental results are used to verify the accuracy of the numerical CFD simulations performed with no SCD.

The multi-element unstructured numerical framework CRUNCH CFD 3.0.0 is used to simulate the inducer flow field over a range of flow coefficients (\(\phi = 0.117, 0.107, 0.097, 0.08, \) and \(0.07\), at an inlet temperature of 21.1 K, and a rotational speed of 30,000 RPM. Flow coefficients
of $\phi = 0.117$ and $\phi = 0.107$ using the inducer with no SCD are compared to experimental data to determine the validity of the numerical simulations.

Prior work has shown that SCD effectiveness in general is limited for $AR < 2.3$, where $AR$ is the area ratio. $AR$ is the ratio of the blade throat area to the relative flow area upstream of the inducer, and is found using Eq. 4.5, where $A_{throat}$ is the throat area of a single blade passage and $A_{inlet}$ is the upstream flow area of a single blade passage. $AR$ describes the single phase flow diffusion as the fluid enters the inducer. For $AR > 2.3$, SCD implementation has been shown to suppress cavitation instabilities and allow for stable operation at flow coefficients far below design [16]. The flow coefficients $\phi = 0.097$ and $\phi = 0.08$ were simulated to test SCD effectiveness below design. No experimental data is available for these flow coefficients, as the lowest flow coefficient tested at 21.1 K by Meng was $\phi = 0.101$. To further increase cavity volumes in the inducer, and potentially highlight the benefits of SCD implementation, a flow coefficient of $\phi = 0.07$ was also simulated. This value of $\phi$ is $\sim 60\%$ of the design flow coefficient and is far lower than any flow coefficient tested experimentally by Meng at 21.1 K [29].

$$AR = \frac{A_{throat}}{A_{inlet}} = \frac{sin\beta_b}{sin\beta} = \frac{sin\beta_b}{sin(atan\phi)}$$

(4.5)

CRUNCH CFD has a variety of solver modules. For this work, the incompressible module was used. The incompressible module neglects density changes and work due to pressure. Instead, density changes are a function of cavitation and fluid property variation due to thermal effects [72]. This provides two distinct advantages to a more general and complete compressible formulation; namely the computational cost is significantly lower, and the simple formation is more robust [72]. The simplification provided by this model is critical to resolve the full 3D flow field of such a complex problem. The steady Reynolds-averaged Navier-Stokes (RANS) equations are solved with the standard high Reynolds number form of the $k$-$\epsilon$ equations to model turbulence. To handle cavitation, a simple finite rate model is employed. A more sophisticated bubbly cavitation model is available in CRUNCH, but produces very similar results to the finite rate model when large scale unsteadiness is not present [72]. As the steady RANS equations are solved in this work, no large scale unsteadiness was present; therefore the finite rate cavitation model was deemed sufficient.
The inducer computational domains are defined by meshes generated in Autogrid. The meshes have $9.5 \times 10^6$ and $12.1 \times 10^6$ cells for the baseline inducer and the inducer with an SCD, respectively, with refinement near the inducer blades in both cases. Cross sections of the inducer meshes with and without the SCD are included in Fig. 4.2. The cell count difference between the two meshes stems from the additional computational domain volume due to the SCD and the additional refinement near the SCD bleed and reinjection slots. The baseline inducer mesh use 6 cells to span the tip gap, and has an average $y^+$ of 68. The inducer with SCD mesh uses 14 cells to span the tip gap, and has an average $y^+$ of 55. The difference in mesh resolution to resolve the tip gap between the two meshes is a function of SCD implementation and best practices for generating a structured mesh. To avoid mass flow imbalances in the transition between rotor and SCD, mesh refinement in the tip gap extends $0.4D_{tip}$ in the axial direction upstream and downstream of the inducer leading edge in the SCD mesh. While mass flow imbalances in the transition from rotor to SCD were removed, the refinement in the tip clearance was increased over the baseline inducer mesh. Note the SCD restriction near the reinjection slot in Fig. 4.2. This is a deliberate feature intended to control the flow velocity through the SCD in an attempt to prevent cavitation growth and surge in the SCD, which can negate benefits of SCD implementation [8].

To determine grid independence, a Richardson based error estimation using a safety factor of 1.25 [75, 76] was performed to determine the grid convergence index (GCI) using additional meshes of $16.2 \times 10^6$ and $1.0 \times 10^6$ cells using the inducer geometry with no SCD. GCI was determined at two points: single phase operation and deep into breakdown, where cavitation formation has reduced head coefficient to 54% of the single phase value. GCI during single phase operation was determined using predicted head coefficient, while GCI during operation in deep breakdown was determined using predicted NPSH. When little to no cavitation is present, accurate prediction of head coefficient is the most important consideration. However, when significant cavity volumes form deep into breakdown, the accurate prediction of NPSH becomes more important. Comparing GCI based on both head coefficient and NPSH gives a better understanding of the solution dependence on grid density across the full breakdown curve. The resulting GCIs are included in Table 5.1 and provide error band estimations in both axes of the breakdown curve. GCI$_{12}$ shows the estimated error of the mid level mesh refinement (mesh 2) when compared to the most refined mesh (mesh 1) . GCI$_{23}$ shows the estimated error of the mid level mesh (mesh 2) when compared
to the least refined mesh (mesh 3). $GCI_{23}$ is greater than $GCI_{12}$ at both operating points because the mesh density difference is much greater between mesh 2 and mesh 3. Mesh 2 contains 9 times as many cells as mesh 3, while mesh 1 contains 1.8 times as many cells as mesh 2. Despite the large mesh density disparities, the error estimations using GCI are reasonably small for both head coefficient and NPSH. This suggests the mid level mesh refinement used in this work is sufficient. Thus, for this work, all reported values are estimations with error bands based on the average of
Table 4.2: Grid convergence index comparison for varying mesh sizes at two operational points. These values loosely represent error bands for data extracted from the numerical simulations.

<table>
<thead>
<tr>
<th></th>
<th>Single Phase $\psi$</th>
<th>Deep Breakdown NPSH</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCI$_{12}$ (%)</td>
<td>3.00</td>
<td>1.72</td>
</tr>
<tr>
<td>GCI$_{23}$ (%)</td>
<td>9.23</td>
<td>8.75</td>
</tr>
</tbody>
</table>

GCI$_{12}$ and GCI$_{23}$. Head coefficient is assumed to have an error band of 6.1% along the breakdown curve. Additionally, an error band of 5.2% is assumed for NPSH values along the breakdown curve. Simulation convergence using each mesh was determined case by case by evaluating the solution monitors of inlet mass flow, outlet mass flow, inlet total pressure, head coefficient, and lateral forces acting on the hub and inducer blades.

4.5 Results

In this section we first validate the simulation method by comparing experimental data to simulation data. Then, to determine the effects of SCD implementation when pumping liquid hydrogen, we examine various aspects of pump performance including suction performance, rotordynamic forces, stability, and backflow formation.

4.5.1 Simulation Validation

CRUNCH non cavitating simulation data for both SCD and no SCD inducer geometries are shown with experimental head-flow data from Meng [29] in Fig. 4.3. However, to benchmark simulation accuracy, only the no SCD data is used. For all simulated flow coefficients, the average error between experimental and CRUNCH simulated head coefficients for the inducer with no SCD is 11%, with a maximum error of 20.8% at $\phi = 0.117$, due to the small magnitudes of $\phi$ at this condition. This suggests the CRUNCH simulation setup provides a reasonable model of the flow field and yields results from which general observations can be made.
Figure 4.3: Non cavitating performance curve of the considered inducer over the range of considered flow coefficients. Experimental results from Meng [29] are compared to CRUNCH CFD results for both SCD and no SCD configurations.

4.5.2 Suction Performance

The data of Fig. 4.3 show that SCD and no SCD non cavitating performance is essentially the same at the considered flow coefficients. The head coefficient produced by the inducer with an SCD is within 7% of the no SCD head coefficient for all values of $\phi > 0.08$, where the no SCD head value is greater than the SCD value. However, at $\phi = 0.07$, the SCD head coefficient is 5% greater than the no SCD head coefficient. This suggests the benefits of SCD implementation seen in cold water are not as prominent in cryogenic fluids, and only occur at very low flow coefficients for inducers operating in the single phase regime.

To understand how the SCD affects inducer suction performance, performance curves characterizing inducer cavitation breakdown are shown in Fig. 4.4. Experimental data for the inducer with no SCD at $\phi = 0.117$ and $\phi = 0.107$ are shown as solid and dashed lines respectively. Overall agreement between the CRUNCH and experimental results is fair; the simulation data profiles are similar to the experimental data profiles, although the magnitude of the experimental results
Figure 4.4: Cavitation breakdown curves for all considered flow coefficients. Experimental results from Meng [29] are compared to CRUNCH CFD results for both SCD and no SCD configurations at $\phi = 0.117$ and $\phi = 0.107$. Meng did not test the inducer at 21.1 K for $\phi = 0.097$ or $\phi = 0.07$.

are slightly higher than the simulation data. At $\phi = 0.117$, CRUNCH under predicts $\psi$ at NPSH $> 200$ by 10.6%. Additionally, the NPSH at which the inducer starts to break down, as predicted by CRUNCH, is within $\sim 20\%$ of the experimental value. A different trend is seen at $\phi = 0.107$. CRUNCH under predicts $\psi$ for NPSH $> 190$ by only 4.5%, which is an improvement over the $\phi = 0.117$ result. However, at this flow coefficient, CRUNCH over predicts the NPSH where breakdown begins by $\sim 23\%$. These are the only flow coefficients that can be compared to experimental data as Meng does not provide cavitation performance curves for $\phi = 0.097$, $\phi = 0.08$, or $\phi = 0.07$ at 21.1 K [29].

Considering modeling limitations, similarity of breakdown curve profiles between experimental data and CRUNCH simulations in Fig. 4.4 is good. Hosangadi [41] noted similar discrepancies between experimental and simulation data for a similar inducer, although he did not consider an SCD. While simulated head coefficient and NPSH magnitudes would surely differ from experimentally obtained results at lower flow coefficients, the CRUNCH simulation data provides a relative comparison between an inducer with and without an SCD. In this way, we can
determine the usefulness of an SCD while considering the thermodynamic effects of cavitation even if simulated values differ slightly from experimental results.

The differences between the SCD and no SCD breakdown curves of Fig. 4.4 are significant. However, both the SCD and no SCD breakdown curves exhibit a gradual breakdown. In this work, breakdown corresponds to a 3% drop in head coefficient from the single phase value [3]. At \( \phi = 0.117 \), head rise produced by the inducer with no SCD is 11% greater than that produced by the inducer with the SCD at high NPSH. As cavity volumes increase, the head performance for inducer with the SCD decreases, and breakdown NPSH is predicted to be \( \sim 250 \) ft. This is slightly lower than the predicted breakdown NPSH of \( \sim 300 \) ft for the inducer with no SCD.

At \( \phi = 0.107 \) the inducer with no SCD produces a head rise 5% greater than that produced by the inducer with an SCD at high NPSH. The predicted breakdown for the inducer with the SCD is \( \sim 215 \) ft. This is again slightly lower than the predicted breakdown NPSH of \( \sim 227 \) ft for the inducer with no SCD. The \( \phi = 0.107 \) and 0.117 flow points are near the design point, and the SCD is not designed to improve performance near the design point. The current data show that indeed no head performance benefit of SCD implementation is apparent near the design point. However, very slight improvements of breakdown NPSH are noted.

A similar trend between the SCD and no SCD head coefficient results is also evident for \( \phi = 0.097 \). At high NPSH for this flow coefficient, the inducer with no SCD produces a head rise \( \sim 6\% \) greater than that produced by the inducer with the SCD. The predicted breakdown NPSH for the inducer with the SCD operating at \( \phi = 0.097 \) is \( \sim 230 \) ft. This is slightly higher than the predicted breakdown NPSH of \( \sim 195 \) ft for the inducer with no SCD at the same \( \phi \).

The head coefficient difference between the SCD and no SCD scenarios at NPSH values above the breakdown point is reduced to \( \sim 1\% \) as flow coefficient drops to \( \phi = 0.08 \). At this flow coefficient, the predicted breakdown NPSH for the inducer with the SCD is \( \sim 190 \) ft. The predicted breakdown NPSH for the inducer with no SCD is slightly higher at \( \sim 285 \) ft. At this flow coefficient the inducer with no SCD again produces the higher head coefficient, but exhibits a higher breakdown NPSH.

As \( \phi \) drops from 0.117 to 0.08, the SCD and no SCD breakdown curves move closer together. Only at a \( \phi \) value of \( \sim 60\% \) of the design flow coefficient (\( \phi = 0.07 \)) is any suction performance improvement due to the SCD evident. This is consistent with SCD implementation
with an inducer pumping cold water [16]. At $\phi = 0.07$, the inducer with the SCD produces a 4.4% higher head coefficient at high NPSH = 384 compared to the inducer with no SCD inducer at NPSH = 514. These two points provide the best comparison of SCD and no SCD performance at $\phi = 0.07$ because any additional points simulated for the inducer with no SCD below NPSH = 500 become unstable and are unable to reach a converged solution. Simulation divergence is likely a function of significant flow instabilities overwhelming the incompressible model incorporated in the simulation. It appears that the SCD stabilizes the flow enough to allow the solver to reach a solution at this very low value of $\phi$. At $\phi = 0.07$, the inducer with the SCD exhibits a predicted breakdown NPSH of $\sim 220$ ft.

Prior work using cold water as the working fluid has found SCD implementation far below the design point to be beneficial in increasing head coefficient over the same inducer with no SCD [16]. This trend continues when liquid hydrogen is the working fluid. Head improvements corresponding to SCD implementation in liquid hydrogen are only evident at very low flow coefficients ($\sim 60\%$ of design). While head performance improvements are possible with an SCD, they are not ubiquitous, and will vary with inducer and SCD geometries.

SCD implementation has also been shown to reduce the breakdown NPSH in an inducer pumping cold water [16]. The breakdown curves presented here with liquid hydrogen as the working fluid are gradual and do not exhibit the sharp knee characteristic of breakdown curves with water as the working fluid [16]. Slight reductions in breakdown NPSH are evident at $\phi = 0.117$, 0.107, and 0.08 for the inducer with an SCD. Again, while breakdown NPSH improvements are possible with an SCD in liquid hydrogen, they are likely not ubiquitous, and will vary with inducer and SCD geometries.

4.5.3 Rotordynamic Forces

SCD implementation has also been previously shown to reduce rotordynamic forces that can lead to structural failure with inducers pumping cold water [16]. This observation continues for SCD implementation in an inducer pumping liquid hydrogen. Figure 4.5 shows the steady solver lateral force coefficient on the inducer as a function of NPSH. For all simulated NPSH at flow coefficients $\phi < 0.117$, the inducer with the SCD exhibited smaller rotordynamic forces than the inducer with no SCD.
Figure 4.5: Lateral force coefficient acting on both inducer geometries for all considered flow coefficients. SCD implementation reduces lateral forces on the inducer for all simulated flow coefficients.

For the inducer with no SCD at $\phi = 0.117$, forces increase with increasing NPSH until $\sim 230$, above which forces gradually decrease. The SCD inducer at $\phi = 0.117$ exhibits a different trend. Here, the lateral force remains relatively constant for NPSH $< 200$, while above this value the force coefficient decreases. As the flow coefficient is decreased to $\phi = 0.107$, the force coefficient exhibits an average increase of 28% for the case with no SCD and 3% average increase for the SCD case. At this flow coefficient, the stabilizing effects of the SCD are becoming evident. Lateral force increases much more rapidly with decreasing $\phi$ at all NPSH values for the inducer with no SCD than for the inducer with the SCD. Overall force coefficient again increases as flow coefficient decreases to $\phi = 0.097$, with the no SCD case increasing by $\sim 23\%$ on average and the SCD case increasing by $\sim 6\%$ on average. As flow coefficient decreases to $\phi = 0.08$, force coefficient increases by an average of $\sim 20\%$ for the inducer with no SCD and $\sim 23\%$ for the inducer with the SCD. Force coefficient again increases as flow coefficient decreases to $\phi = 0.07$. With this drop in flow coefficient, force coefficient increases by an average of $\sim 11\%$ for the inducer with no SCD and $\sim 8\%$ for the inducer with the SCD.
Decreasing $\phi$ causes consistently large ($20\% - 30\%$) increases in force coefficient for the inducer with no SCD, with the exception of $\phi = 0.07$. Force coefficient increases caused by decreasing $\phi$ are consistently small ($3\% - 8\%$) for the inducer with the SCD, with the exception of $\phi = 0.08$. In general, the force coefficient curves for the inducer with no SCD at all $\phi$ values exhibit similar profiles that asymptote to a specific value with increasing NPSH. However, the SCD force curves generally show a more parabolic trend for which force coefficient can decrease with increasing NPSH above a specific NPSH value.

### 4.5.4 Stability

Figures 4.6 to 4.10 show profiles of the flow at the leading edge of the blade. Local incidence values are shown as a function of blade span. Profiles for the inducer with and without the SCD are provided at the highest and lowest simulated NPSH scenarios. Incidence is defined in Eqn. 4.6, where $\beta_b$ is the inlet blade angle measured from the tangential direction and $\beta$ is the inlet flow angle measured from the same direction. For axial flows, $\beta$ is the arctangent of the flow coefficient in a bulk flow analysis. Incidence is a measure of blade loading and can greatly effect pump performance [6]. Figures 4.6 to 4.10 provide an overview of how incidence changes along the blade span as the inducer breaks down for all simulated flow coefficients.

$$i = \beta_b - \beta \quad (4.6)$$

Figure 4.6 provides the local incidence for the highest and lowest simulated NPSH values at $\phi = 0.117$. For the inducer with the SCD, high NPSH is 482 ft and low NPSH is 136 ft. For the inducer with no SCD, high NPSH is 650 ft, and low NPSH is 130 ft. Both the SCD and no SCD cases exhibit essentially constant incidence at $2^\circ$ from $10\% - 40\%$ span at low NPSH. Above $40\%$ span, the incidence increases from $\sim 2^\circ$ to $\sim 6^\circ$ for the no SCD case, while incidence only increases to $\sim 3^\circ$ for the SCD case. At high NPSH, incidence for the SCD case is essentially the same as low NPSH from $10\% - 75\%$ span, but increases by $0.9^\circ$ on average from $80\% - 100\%$ span, with a maximum increase of $1.36^\circ$ at $100\%$ span. Conversely, incidence for the no SCD case at high NPSH follows the same trend seen at low NPSH, but is on average $1.46^\circ$ greater at all span locations, with a maximum increase of $5.14^\circ$ at $100\%$ span. The lower incidence corresponding
Figure 4.6: Incidence along blade span at $\phi = 0.117$ for the highest and lowest simulated values of NPSH.

to SCD implementation results in the lower head coefficient at $\phi = 0.117$ in Fig. 4.4. Head rise in helical inducers is caused by incidence effects, with the incidence at the tip being the primary factor. As the SCD reduces incidence from 40%–100% span, the blade loading decreases, reducing the overall head rise.

As the flow coefficient is decreased to $\phi = 0.107$, the local incidence values change significantly for the no SCD case. Figure 4.7 plots local incidence along blade span for the highest and lowest simulated NPSH values at $\phi = 0.107$. For the inducer with the SCD, high NPSH is 278 ft and low NPSH is 100 ft. For the inducer with no SCD, high NPSH is 316 ft, and low NPSH is 101 ft. At low NPSH, the incidence for the inducer with no SCD decreases from $\sim 6^\circ$ at 10% span to $\sim 4^\circ$ at 90% span. From 90% to 100% span, incidence increases rapidly to $\sim 12^\circ$. In contrast, at low NPSH, the SCD case again maintains relatively constant incidence of $\sim 2^\circ$ from 10%–75% span. From 75%–100% span, incidence increases to $\sim 4^\circ$. This same trend continues for the SCD case at high NPSH, but with a 0.5° average increase from 65%–85% span. The local incidence for the no SCD case at $\phi = 0.107$ is 0.86° greater between 10% and 80% span at high NPSH than
Figure 4.7: Incidence along blade span at \( \phi = 0.107 \) for the highest and lowest simulated values of NPSH.

at low NPSH. The maximum difference between the low and high NPSH incidence values for the no SCD case at this flow coefficient is 1.1°, and occurs at 10% span. Incidence again increases rapidly above 90% span to reach a maximum incidence of \( \sim 15^\circ \) at 100% span at high NPSH with no SCD.

Trends in the local incidence are similar as the flow coefficient decreases to \( \phi = 0.097 \). Figure 4.8 plots incidence along blade span for the highest and lowest simulated NPSH values at \( \phi = 0.097 \). For the inducer with the SCD, high NPSH is 432 ft and low NPSH is 66 ft. For the inducer with no SCD, high NPSH is 281 ft, and low NPSH is 72 ft. At low NPSH, local incidence for the inducer with no SCD decreases from \( \sim 7^\circ \) at 10% span to 4.2° at 85% span. Incidence then increases to 14.8° at 100% span. As with the \( \phi = 0.117 \) and 0.017 cases, the SCD continues to maintain a local incidence of \( \sim 2^\circ \) for the majority of the span at low NPSH when \( \phi = 0.097 \). Incidence remains around 2° below 75% span, but then increases to reach a maximum of 12.6° at 100% span. At high NPSH, incidence for the SCD case increases by 0.44° on average from 10% – 45% span over the low NPSH values. Between 50% and 80% span, incidence increases by
0.8° on average over the low NPSH values, with a maximum increase of 1° occurring at 65% span. At high NPSH, incidence for the inducer with no SCD follows the same trend seen at low NPSH, but is on average 1° greater at all span locations, with a maximum increase of 3° at 95% span.

Figure 4.9 plots local incidence for the highest and lowest simulated NPSH values at φ = 0.08. For the inducer with the SCD, high NPSH is 440 ft and low NPSH is 30 ft. For the inducer with no SCD, high NPSH is 591 ft, and low NPSH is 9 ft. At this flow coefficient, the trends in incidence trends vary significantly from those at higher flow coefficients. At low NPSH, incidence for the no SCD case decreases from ∼7° at 10% span to ∼4° at 90% span, and increases to ∼14° at 100% span. However, at high NPSH, incidence for the no SCD case decreases from ∼8.5° at 10% span to ∼6° at 40% span. Incidence begins to increase at 45% span, to reach a maximum of 19.2° at 100% span. At low NPSH the SCD case again maintains low, consistent incidence, with an average of 2.5° between 10% and 95%, but increases to ∼9° at 100% span. At high NPSH, average incidence is ∼3° from 10% – 35% span, but increases to maintain an average of

Figure 4.8: Incidence along blade span at φ = 0.097 for the highest and lowest simulated values of NPSH.
Figure 4.9: Incidence along blade span at $\phi = 0.08$ for the highest and lowest simulated values of NPSH.

$3.4^\circ$ between 50% and 75% span. Incidence then increases, to reach a maximum of $14.9^\circ$ at 100% span.

The greatest difference in incidence between SCD and no SCD cases is seen in Fig. 4.10, which plots local incidence for the highest and lowest simulated NPSH values at $\phi = 0.07$. For the inducer with the SCD, high NPSH is 384 ft and low NPSH is 65 ft. For the inducer with no SCD, high NPSH is 627 ft, and low NPSH is 514 ft. Incidence varies very little with NPSH for the inducer with no SCD, with an average difference between the two curves of $0.02^\circ$ along the whole span. Incidence decreases from $\sim 8^\circ$ at 10% span to $\sim 7^\circ$ at 40% span, then gradually increases to $9.8^\circ$ at 90% span, and finally increases sharply to $\sim 20^\circ$ at 100% span. The incidence for the SCD case exhibits some small variation with NPSH. At low NPSH, incidence is relatively constant and averages to $3.2^\circ$ from 10% to 40% span. Incidence then increases to $3.6^\circ$ at 60% span, decreases to $3.4^\circ$ at 80% span, and increases to reach a maximum of $14.6^\circ$ at 100% span. At high NPSH, incidence from 10% to 40% is $2.5^\circ$ on average. Incidence then increases gradually to $\sim 4.5^\circ$ at 75% span, and then increases sharply to $15.4^\circ$ at 100% span. At this flow coefficient, it is clear
that the SCD significantly reduces incidence, resulting in an average reduction of 4.5° through the span.

To easily compare local incidence values between the SCD and no SCD cases, Table 4.3 provides local incidence values for all simulated flow coefficients averaged along the blade span and averaged along all NPSH values. Across all flow coefficients, the incidence for the inducer with the SCD is 3.5° less on average than the incidence for the inducer with no SCD.

Table 4.3 also helps determine the relationship between incidence and force coefficient. The greatest average difference in incidence for the inducer with an SCD occurs when flow coefficient decreases from \( \phi = 0.097 \) to \( \phi = 0.08 \). This decrease in flow coefficient results in an average incidence increase of 0.78° over all NPSH values. The same decrease in flow coefficient from \( \phi = 0.097 \) to \( \phi = 0.08 \) results in an average force coefficient increase of 23% on average over all NPSH values, which can be seen in Fig. 4.5. The inducer with no SCD maintains the same relationship between incidence increases and force coefficient increases. Average incidence increases by a maximum of 1.86° when flow coefficient decreases from \( \phi = 0.117 \) to \( \phi = 0.107 \).
Force coefficient increases by an average of 28% over all NPSH values for this same decrease in flow coefficient (Fig. 4.5). The relatively large increases in incidence due to decreases in flow coefficient result in relatively large increases in force coefficient.

For all flow coefficients that were considered, local incidence at the leading edge is lower for the inducer with an SCD than without. Lower incidence decreases the loading on the front of the blade, and results in lower lateral force when the inducer uses an SCD. It is evident that the stabilizing effect of the SCD observed with water as the working fluid [8] extends to liquid hydrogen. The stabilizing effect results from controlling incidence such that it remains low (2 – 5°) despite large decreases in flow coefficient. Even at low flow coefficients, which typically exhibit large flow angles and incidence with no SCD (Fig. 4.10), SCD implementation maintains consistent low incidence on the blades at flow coefficients far below the design condition with liquid hydrogen as the working fluid.

By extracting high pressure fluid just downstream of the leading edge and reintroducing it upstream of the inducer, the SCD increases local mass flow at the leading edge of the blades, increasing the flow angle. An important parameter is the SCD mass flow gain factor, $K_{SCD}$. $K_{SCD}$ describes the increased mass flow at the leading edge of the inducer due to the rejetted flow and is defined in Eq. 4.7. $\dot{m}_{SCD}$ is the mass flow through the SCD and $\dot{m}_{in}$ is the inlet mass flow. Another important parameter is the cavitating area ratio, $AR_c$. The cavitating area ratio accounts for the reduced throat area causes by cavitation blockage, and is defined in Eq. 4.8 by multiplying the single phase area ratio by a corrective term $1 - T_c$ [55]. $T_c$ the area average of the volume fraction of vapor over the throat area, and describes the throat area blocked by cavitation. $T_c$ is

### Table 4.3: Local incidence averaged along span and all NPSH values for all simulated flow coefficients

<table>
<thead>
<tr>
<th>$\phi$</th>
<th>Average Incidence (°)</th>
<th>No SCD</th>
<th>SCD</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.117</td>
<td>4.3</td>
<td>2.3</td>
<td></td>
</tr>
<tr>
<td>0.107</td>
<td>6.1</td>
<td>2.8</td>
<td></td>
</tr>
<tr>
<td>0.097</td>
<td>7.0</td>
<td>3.3</td>
<td></td>
</tr>
<tr>
<td>0.08</td>
<td>7.7</td>
<td>4.0</td>
<td></td>
</tr>
<tr>
<td>0.07</td>
<td>9.1</td>
<td>4.7</td>
<td></td>
</tr>
</tbody>
</table>
defined in Eq. 4.9, where $\phi_{vapor}$ is the volume fraction of vapor. The cavitating area ratio provides a measure of the inlet diffusion of an inducer operating under cavitation conditions. Inlet diffusion is an important parameter in describing inducer stability because a stall occurs at the inlet of the inducer and causes upstream backflow as inlet diffusion increases [16, 55].

$$K_{SCD} = 1 + \frac{\dot{m}_{SCD}}{\dot{m}_{in}} \quad (4.7)$$

$$AR_c = AR \times (1 - T_c) \quad (4.8)$$

$$T_c = \frac{\int \phi_{vapor} dA}{A_{throat}} \quad (4.9)$$

Figure 4.11 plots $K_{SCD}$ as a function of $AR_c$, and helps to explain why the SCD has a stabilizing effect. The SCD mass flow gain factor increases linearly with increasing $AR_c$, following the relation $K_{SCD} = 0.374 \times AR_c + 0.579$, which provides a useful reduction in incidence as inlet diffusion and flow instabilities increase. This is the reason incidence with the SCD remains within $\sim 2$ degrees of the design point value when operating at $\phi = 0.07$. The linear increase in $K_{SCD}$ is a function of cavity formation in liquid hydrogen, as cavity volumes contain far less vapor and spread out more than in water [40, 41, 43], forming a gradient of vapor rather than distinct structures evident in water.

4.5.5 Backflow Formation

The improved stability, due to consistent low values of incidence that result from SCD implementation and mass flow gain is important and reduces the amount of backflow and removes effects of the tip vortex that typically forms at the leading edge of the inducer [16, 55]. Backflow and tip vortex formation are functions of flow diffusion as fluid enters the inducer, also known as inlet diffusion [55]. Figure 4.12 shows the dimensionless backflow upstream penetration, $L_p/D_{tip}$, as a function of $AR_c$. At $\phi = 0.117$, very little backflow is evident and both SCD and no SCD geometries exhibit approximately the same upstream backflow penetration. Low backflow at low $AR_c$ is expected, and Japikse predicted a critical area ratio for which an inducer will begin to
Figure 4.11: SCD mass flow gain factor as a function of $AR_c$ for all simulated flow coefficients. A curve fit of the form $K_{SCD} = 0.3737 \times AR_c + 0.5789$ is also shown.

develop backflow to be around 1.5 [19, 20]. As evident in Fig. 4.12, backflow penetration for the inducer with no SCD increases sharply for $1.3 < AR_c < 1.5$, and continues to increase at a constant lesser rate for $AR_c > 1.5$. The point at which backflow begins to develop is $\sim 1.3$, and is therefore defined as critical $AR_c$ for the inducer with no SCD pumping liquid hydrogen considered here. Backflow penetration for the inducer with the SCD remains relatively constant for $1.3 < AR_c < 1.5$. Backflow sharply increases for $1.5 < AR_c < 1.6$, and continues to increase at a constant lesser rate for $AR_c > 1.6$. The critical $AR_c$ for the inducer with an SCD pumping liquid hydrogen is therefore 1.5, which is the point where backflow begins to develop. The higher critical $AR_c$ corresponding to the SCD case is again a function of the controlled incidence at the leading edge and means that an inducer with an SCD can operate at a lower flow coefficient (higher inlet diffusion), while experiencing less backflow than the same inducer with no SCD. In this case, the inducer with an SCD exhibits essentially constant backflow upstream penetration for both $\phi = 0.117$ and $\phi = 0.107$. 

98
Figure 4.12: Upstream backflow penetration plotted as a function of $AR_c$, which considers the reduction in flow area due to cavitation blockage. The SCD significantly limits backflow penetration at all flow coefficients below $\phi = 0.117$.

Note the y-axis of Fig. 4.12 is logarithmic. Backflow growth follows an exponential curve with increasing $AR_c$ for both SCD and no SCD cases. However, backflow grows at a greater rate in the inducer with no SCD. Overall magnitudes of backflow upstream penetration are relatively small, with backflow propagating up to $\sim 1.5D_{tip}$ upstream at $\phi = 0.07$ with no SCD. Other inducers operating in water exhibited backflow upstream penetration on the order of $6D_{tip}$ upstream of the leading edge [55]. While the backflow growth presented in Fig. 4.12 follows an exponential curve, it is rather gradual. The gradual formation of backflow is due to the nature of cavitation in a cryogenic fluid. Fluids at cryogenic temperatures are defined by thermal effects caused by low liquid to vapor density ratios. The low liquid to vapor density ratios impart a high dependence of vapor pressure on temperature, which results in spread out and porous cavities forming in liquid hydrogen [40, 41, 43]. As the porous cavity volumes form and spread through the blade passages, relatively small of the throat are blocked when compared to the exponential growth of cavitation in water. The marginal blockage of the throat causing a gradual reduction of $AR_c$ rather than a steep decrease corresponding to the exponential cavity growth in cold water [55] which results in gradual
backflow growth. However, the tendency of SCD implementation to reduce backflow formation and growth [16] is maintained with a cryogenic working fluid, as SCD implementation suppresses backflow formation as $\phi$ falls below design.

Figure 4.13 plots $AR_c$ as a function of cavitation number, and helps describe the shape of the breakdown curves in Fig. 4.4 and the backflow curves in Fig. 4.12. As can be seen in Fig. 4.4, breakdown for the considered inducer is relatively gradual. Additionally, backflow increases at an essentially linear rate with increasing $AR_c$. Both relationships are explained by the nature of cavitation in liquid hydrogen. As seen in Fig. 4.13, as cavitation number decreases, $AR_c$ remains relatively constant for all simulated flow coefficients. Cavitation growth and flow blockage is hindered by TSH, which results in minimal reductions in $AR_c$ with large decreases in cavitation number. This results in the gradual breakdown and linear increases in backflow previously discussed.

4.6 Conclusions

This work explored the effects of implementing an SCD in cryogenic liquid hydrogen. CRUNCH CFD simulation data was shown to agree reasonably well with experimental data for
non-cavitating inducer performance. When modeling cavitation, the simulation results for head coefficient deviated only modestly from experimental data. The simulations allow investigation of the inducer flow field and performance while considering the thermodynamic effects of cavitation in liquid hydrogen at cryogenic temperatures.

Other work has shown SCD implementation can improve pump performance with water as the working fluid, specifically with increases in head coefficient at flow coefficients below design. With liquid hydrogen as the working fluid, no significant head improvements resulting from SCD implementation were apparent at the simulated flow coefficients. However, SCD implementation was found to increase inducer stability by maintaining low incidence values at all simulated flow coefficients. Low incidence values caused by SCD implementation were also found to limit backflow formation and upstream propagation.

In liquid hydrogen, the inducer with no SCD produced up to 43% greater head than the inducer with an SCD at NPSH values in cavitation breakdown. It is only at low flow coefficients (∼60% of design) that the SCD appears to provide head improvements when pumping liquid hydrogen. At ∼60% of the design flow coefficient, the SCD provides an increase in head coefficient of 4.4% over the no SCD. Importantly, at this flow point, the flow field for the inducer with no SCD is so unstable that the simulations are unable to reach a converged solution. This is in contrast to simulations with the SCD, where the flow remains stable enough for the solver to continue simulating the flow.

SCD implementation also affected the breakdown NPSH of the inducer. When breakdown is defined as a 3% reduction in head coefficient from the non-cavitating head coefficient, the predicted breakdown NPSH was lower for the inducer with the SCD operating at φ = 0.117, 0.107, and 0.08. The SCD reduced breakdown NPSH by 16.7% at φ = 0.117, 5.3% at φ = 0.107, and 33.0% at φ = 0.08.

While the SCD provides no real benefit in head coefficient, significant improvements in inducer stability were noted. The increased stability is a function of low incidence over the range of simulated flow coefficients, caused by increased mass flow at the blade leading edge. This is due to the mass flow gain from the SCD reinjection slots. The mass flow gain caused by the SCD was shown to increase linearly with inlet diffusion allowing for a greater reduction in incidence at lower flow coefficients where incidence naturally increases due to decreasing flow angles. The low
incidence maintained by the SCD at flow coefficients far below design also limits flow diffusion and stall at the blade inlets. The limited flow diffusion in turn results in reduced backflow upstream penetration. All the benefits of SCD implementation in liquid hydrogen are a result of limiting incidence at low flow coefficients.

At all flow coefficients below $\phi = 0.117$, the SCD significantly represses backflow formation which provides myriad benefits in a pump, including a reduction in cavitation instabilities due a stabilized upstream flow field. The reduction in backflow formation decreases structural forces on the inducer at all flow coefficients below $\phi = 0.117$, and is more apparent at flow coefficients far below design.

While many of the benefits noted from SCD implementation in cold water also apply to implementation in liquid hydrogen, the most important contribution of SCD implementation in liquid hydrogen is maintaining a low average incidence through a wide range of flow coefficients. While the low incidence limits the potential head produced by the inducer, this is offset by multiple benefits. Low incidence reduces cavitation formation, significantly improves inducer stability, and limits backflow formation. SCD implementation allows an inducer to operate stably without backflow over a wider range of flow coefficients than is possible with no SCD, even with liquid hydrogen as the working fluid.
CHAPTER 5. CAVITATION INCEPTION AND PERFORMANCE OF A CENTRIFUGAL IMPELLER DURING START-UP

This chapter is a paper to be submitted to the Journal of Fluids Engineering of the American Society of Mechanical Engineers (ASME). The formatting of this paper has been modified to meet the stylistic requirements of this dissertation.

5.1 Contributing Authors and Affiliations

Tate Fanning, Steven Gorrell, Daniel Maynes, Department of Mechanical Engineering, Brigham Young University, Provo, UT 84602, USA. Kerry Oliphant, Concepts NREC, White River Junction, Vermont 05001, USA.

5.2 Abstract

Rapid acceleration of rocket engine turbopumps during start-up imparts significant transient effects to the resulting flow field, causing pump performance to vary widely when compared to quasi-steady operation. To help improve turbopump design in response to the transient effects of start-up this paper presents a method to simulate turbopump start-up using CFD. Cavitating pump performance is initially evaluated using a simulation with a constant outlet pressure boundary condition. Based on the difference between simulation inlet pressure and target inlet pressure, the defined pressure on the outlet boundary condition is modified. This process is repeated until simulation inlet pressure is essentially constant during start-up. Using this novel simulation method, the performance of a centrifugal turbopump during start-up is simulated. Reasonable solution convergence is reached in one single phase and four cavitating simulation iterations. After these five simulation iterations, the average difference between inlet pressure and inlet target pressure is 10%. Simulation iterations 4 and 5 agree within 11% on average for inlet total pressure during startup, 0.1% on average for head coefficient, 13% on average for cavitation volume, 20%
on average for flow coefficient, and 2\% on average for RMS force on the impeller. The agreement
between simulation iterations 4 and 5 suggests that a reasonable solution has been reached.

5.3 Introduction

Cavitation inception and the resulting effects on pump performance are typically consid-
ered during quasi-steady operation. This method of analysis can capture cavitation formation and
instabilities as well as performance of the pump for a constant rotational rate, which is typical
operation for most turbopumps. However, several applications require controlled operation from
start-up to shutdown [91]. These unsteady operational parameters are characterized by rapidly
increasing rotation rate, which generates substantial pressure fluctuations that cause additional
transient effects in the formation and growth of cavitation that are not evident during quasi-steady
operation. These additional transient effects during start-up are not insignificant and are governed
mainly by the increasing rotational speed \( \frac{d\omega}{dt} \) and the flow rate increase \( \frac{dQ}{dt} \) [92].

Various aspects of transient flow in centrifugal pumps have been explored for quite some
time. Lefebvre measured the performance of a centrifugal pump during quasi-steady operation in
addition to testing four different start-up profiles [91]. He found transient head to be consider-
ably higher than the quasi-steady operation at the start of the transient operation due to impulsive
pressure rise. However, head dropped and remain below the quasi-steady value for the rest of
the start-up as the effects of the impulsive pressure decay. Tsukamoto found similar results in
his experimental and theoretical study, and concluded that the period of head depression below
quasi-steady values was due to the lag of circulation growth around the impeller vanes [93].

Additional efforts have further explored the transient start-up of an impeller experi-
mentally [62–65]. Some have found that non-dimensional head is very high at the beginning of start-
up, but falls rapidly and recovers to a final, stable value [66]. Others have investigated the response
of of an impeller to sinusoidal variations in rotational speed and found the transient characteristics
of the flow to deviate remarkably from the quasi-steady values [67]. Pressure and flowrate osc-
cillations during transient start-up have been found to be generated by oscillating cavitation [68].
Transient behavior has been found to be caused by oscillating cavitation or water hammer with
water column separation [69], that can potentially cause the complete sudden collapse of the cav-
ity volume [94], with low frequency, high amplitude pressure oscillations occurring at high flow
rates and a pressure decrease at the end of start-up occurring at lower flow rates [70]. Further work has been done that found the backflow region to exhibit a much lower extension during start-up than during steady state operation [71]. In addition to the body of experimental work, models to predict pump performance during cavitating fast start-ups have been developed. Dazin developed a model for the fast start up for incompressible flow turbomachinery operating under non-cavitating conditions. Dazin’s model is based on the angular momentum and energy equation to predict the internal torque, power, and impeller head. The predicted pump head for this model is defined by Eq. 5.1, where $K_1$ and $K_2$ are constant parameters which depend on the geometry of the pump [92].

$$\Delta P = \Delta P_{\text{static}} + \rho K_1 \frac{\partial \omega}{\partial t} - \rho K_2 \frac{\partial Q}{\partial t}$$ (5.1)

Duplaa modified the steady $\Delta P_{\text{static}}$ term of Dazin’s model to adapt the head prediction to fast start up in cavitating conditions. Duplaa used a bilinear interpolation of experimental head drops at multiople flow rates to evaluate a stationary pressure drop, which is then subtracted from Dazin’s $\Delta P_{\text{static}}$ term [95]. Duplaa calls the steady term modified for cavitation development $\Delta P_{\text{static,cav}}$, which he then applies with the density evolution in the impeller to create the variable density head prediction model provided in Eq. 5.2 [95].

$$\Delta P_{\text{cav}} = \frac{\rho(t)}{1000} \Delta P_{\text{static,cav}} + \rho K_1 \frac{\partial \omega}{\partial t} - \rho K_2 \frac{\partial Q}{\partial t}$$ (5.2)

DiMatteo used the commercial tool Ecosim-Pro and the European Space Propulsion System Simulation Library (ESPSS) to simulate the transient performance of a currently available liquid rocket engine [96]. This software package uses NIST tables to model cryogenic fluid properties and cavitation. DiMatteo’s 1-D turbopump used performance maps consisting of data tables from the engine developer to determine head and resistive torque.

These previous modeling studies have focused on experimental work to develop relatively simple models to describe the performance of a specific centrifugal impeller operating under specific conditions during fast start up. Experimental data is exact, but studies are expensive and it is difficult to change impeller geometry quickly and easily. Theoretical models are useful, but can only be applied in specific situations and do not necessarily predict the 3-D intricacies of a flow field generated by a novel impeller geometry. The purpose of this work is to simulate cavitation-
ing performance of a centrifugal impeller during fast start up using computational fluid dynamics (CFD). Fully annular simulation of cavitating performance of a centrifugal impeller during start-up using CFD is a novel effort that offers results specific to the pump design that describe the complete flow field while providing potentially more accurate results than a general theoretical model and being less expensive than a complete experimental study.

5.4 CFD Modeling

A centrifugal impeller is the subject of numerical simulations here. The simulations were performed using the commercial software Star-CCM+ v11.04.010 which has been found capable of accurately modeling cavitation in similar pump systems [16, 55].

5.4.1 Spatial Discretization

The geometry used in simulations here consists of a full annulus inlet pipe, centrifugal impeller and radial outlet. No volute design was considered here, as the main concern of this work was the simulation method rather than actual pump performance. However, this work does investigate the effect of cavitation formation just upstream of the impeller during fast start up. The pump computational domain is defined by a mesh generated in Star-CCM+. The mesh is comprised of $16.6 \times 10^6$ polyhedral cells, and a cross section of the full mesh is provided in Fig. 5.1. To better capture flow physics around the blade tips, refinement increases along the inlet pipe and near the impeller blades and hub, as seen in Fig. 5.2. The average $y^+$ value for the mesh is 73.

To test grid independence, a Richardson based error estimation using a safety factor of 1.25 [75,76] was performed using quasi steady simulation results to determine the grid convergence index (GCI) using additional meshes of $8.94 \times 10^6$ and $25.5 \times 10^6$ cells. The resulting GCIs are included in Table 5.1 and provide an error band estimation for quasi steady pump operation. GCI$_{12}$ shows the estimated error of the base mesh when compared to the more refined mesh. GCI$_{23}$ shows the estimated error of the base mesh when compared to the less refined mesh. Both error estimations using GCI are reasonably small, and indicate the baseline mesh is sufficiently refined and more refined meshes are not required to obtain reasonable simulation accuracy. Thus, for this work, all reported values are estimations with an error band of 2.3%.
Figure 5.1: Mid-plane section of the mesh

Figure 5.2: Tip detail of a mid-plane section of the mesh
Table 5.1: Grid convergence index comparison for three mesh sizes. These values loosely represent error bands for data extracted from the numerical simulations.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>GCI_{12}</td>
<td>0.62</td>
</tr>
<tr>
<td>GCI_{23}</td>
<td>2.31</td>
</tr>
</tbody>
</table>

5.4.2 Cavitation Model

A simple constant density equation of state model is used to describe single phase flow. Simulating pump performance in the single phase regime is done to provide pressure and velocity fields as initial conditions for cavitating simulations. An Eulerian multiphase volume of fluid method [60] is used to describe cavitation formation and flow with liquid water as the primary phase, and water vapor as the secondary phase. This cavitation modeling method uses a simplification of the general Rayleigh-Plesset equation and solves the single phase governing equation set for an equivalent fluid with physical properties defined as functions of constituent phases and volume fractions. Individual cavitation bubbles are not modeled with this approach. A Rayleigh-Plesset formulation that includes the influence of bubble growth acceleration along with viscous and surface tension effects is employed to model rate of vapor production.

In the volume of fluid model, a single set of momentum and turbulence equations are solved to find the distribution of the continuous phase. The dispersed phase is then modeled with a transport equation for the volume fraction [60]. The density and dynamic viscosity are calculated as functions of the physical properties of the constituent phase and its volume fraction.

The governing equations for momentum in the continuous phase and transport in the dispersed phase are described by Eqs. 5.3 and 5.4, respectively.

\[
\frac{\partial}{\partial t}\left(\rho U_j\right) + \frac{\partial}{\partial x_i}\left(\rho U_i U_j\right) = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + \rho g_j + F_j
\]  \hspace{1cm} (5.3)

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0
\]  \hspace{1cm} (5.4)
Where $t$ is time, $U$ is velocity, $P$ is pressure, $g$ is acceleration due to gravity, $F$ is the external force per unit volume, $\mu$ is the dynamic viscosity and $\rho$ is the density. The density and dynamic viscosity are calculated as functions of the physical properties of the constituent phase and its volume fraction, defined in Eqs. 5.5 and 5.6, where $\alpha$ is the relevant volume fraction and is defined by Eq. 5.7.

$$\rho = \sum_i \rho_i \alpha_i$$  \hspace{1cm} (5.5)

$$\mu = \sum_i \mu_i \alpha_i$$  \hspace{1cm} (5.6)

$$\alpha_i = \frac{V_i}{V}$$  \hspace{1cm} (5.7)

### 5.4.3 Turbulence Model

The realizable $k-\varepsilon$ model is used here to model turbulence. This model offers mesh flexibility by applying wall functions to model the boundary layer in the viscous sublayer if the wall $y^+ > 30$, and assumes the mesh density properly resolves the viscous sublayer for regions where the wall $y^+ < 30$. Despite the relatively large overall mesh size, the average $y^+$ of the mesh used here is not necessarily ideal, and wall functions were applied by the turbulence model to help offset the relative coarseness of the mesh to properly resolve the boundary layer.

The governing equations for turbulent kinetic energy, and turbulent dissipation rate, are described by Eqns. 5.8 and 5.9, respectively.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k U_j) = - \frac{\partial P}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \varepsilon - Y_M + S_k$$  \hspace{1cm} (5.8)

$$\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon U_j) = - \frac{\partial P}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon \varepsilon}} + C_1 \varepsilon C_3 \rho \varepsilon + S_\varepsilon$$  \hspace{1cm} (5.9)
Where $k$ is the turbulent kinetic energy, $\varepsilon$ is the turbulent dissipation rate, $\mu_t$ is the turbulent viscosity, $P_k$ is the turbulent production, $P_b$ is the buoyancy production, $\nu$ is the kinematic viscosity, $Y_m$ is the dilation dissipation, and $\sigma_k, \sigma_\varepsilon, S_k, C_1, S_\varepsilon, C_2, C_{1\varepsilon}, C_{3\varepsilon}$ are constants.

Lundgreen modeled the flow physics of a similar computational setup of an axial inducer with a 7° tip blade angle during quasi-steady operation using the same turbulence and cavitation models with good agreement when compared to experimental data, predicting the cavitation number where head breakdown occurs within 1% of the experimentally observed value [16].

### 5.4.4 Boundary and Initial Condition Specification

The computational domain consists of two regions, joined by an internal interface boundary which is visible in Fig. 5.1. This boundary is located $2.8 \times D_{tip}$ upstream of the leading edge of the impeller and separates the inlet pipe region with no rotating components and the rotor region, which contains surfaces that change rotational rate based on time.

Quasi-steady pump performance is often estimated with CFD simulations characterized by a constant rotation rate. A constant rotation rate is modeled in Star-CCM+ with a rotating reference frame or rigid body motion [57]. Rigid body motion moves the mesh cell vertices a fixed displacement per time step, and must be employed in a transient analysis. While this method is the most accurate approach to simulate impeller rotation, it can be very computationally expensive for a large mesh [57], like the one generated for this work. While less accurate, a rotating reference frame is less computationally intensive and provides a compromise between simulation accuracy and computational cost. For this reason, this work employs a rotating reference frame to simulate rotation. The rotor surfaces, which are defined as no slip walls, are the only parts defined to be in the rotating reference frame, and are the only parts subject to rotation in the computational domain. All other boundaries in the domain are defined to be in the lab reference frame, which is stationary.

During start up for this pump, rotation rate changes rapidly from 0 rpm to $\sim 1400$ rpm. The start-up curve defining rotation rate during start up is shown in Fig. 5.3. A user-defined field function is used to generate a spline that interpolates between each point in the curve of Fig. 5.3. This custom field function defines the rotation rate of the rotating reference frame in time.

A mass flow inlet condition is specified for the inlet surface. Both the volume fraction of the phases and mass flow are defined on this type of boundary condition. The inlet is simply defined
Figure 5.3: Impeller rotation rate during start-up for the considered pump

to be fully liquid, but the mass flow is somewhat more complex. As this impeller changes rotation rate with time, inlet mass flow must also change with time as flow through the pump increases with increasing rotational rate. A table defining mass flow in increments of 0.0002 s is input to the inlet boundary condition and defines the mass flow rate profile. Star-CCM+ automatically generates a spline to interpolate between discrete table values to define inlet mass flow at any point in the considered time range.

The radial outlet surface is defined as a pressure outlet. This type of boundary condition requires a static pressure definition as well as a volume fraction definition. As with the inlet boundary, the volume fraction at the outlet is defined to be fully liquid. While pressure outlets define pressure on the outlet boundary, they are also used to control inlet boundary pressure. The outlet pressure less the head rise produced by the impeller then becomes the inlet pressure. As the considered impeller is designed to operate at a constant inlet total pressure during start-up, the defined static pressure at the outlet boundary must vary with time as pump head, which dictates the difference between inlet and outlet boundary pressures, is dependent on the transient rotational rate.

To determine how outlet static pressure must change with time to maintain a constant inlet total pressure, a non-cavitating simulation with constant outlet pressure is run to generate pressure
and velocity field estimates for the full computational domain. The non-cavitating constant outlet pressure simulation data provides inlet total and outlet static pressure curves through the start-up process. The difference between inlet total pressure and target inlet total pressure then acts as a map for how outlet static pressure must change to maintain a constant inlet total pressure.

A table consisting of outlet pressure modified by the difference between actual inlet total pressure and target inlet total pressure in time is used to define the outlet boundary condition of a new simulation that also models cavitation formation. As with the inlet mass flow boundary, Star-CCM+ is able to generate a spline to interpolate between outlet static pressure table values to define outlet pressure at any point in time. After completion of the cavitating simulation with varying outlet pressure, the resulting inlet total pressure data is used to generate another table of revised outlet pressures. Continuing to revise the static pressure data used to define the outlet boundary condition further improves inlet total pressure consistency through time. The process of iterating through simulations to refine the outlet static pressure based on the inlet total pressure data of the previous simulation is outlined in Fig. 5.4. This iterative process is completed four times in total for this work, with inlet total pressure continuing to approach the constant target value with each iteration. For ease of identification, each simulation performed for this work is named. Iteration 1 corresponds to the non-cavitating simulation with constant outlet pressure. Iterations 2 – 5 are all cavitating simulations with varying outlet static pressure. For Iterations 2 – 5, outlet static pressure refinement improves with increasing iteration number.

5.4.5 Processing

To capture the transient flow physics generated by transient start up, all simulations must employ an unsteady solver. Therefore, a time step, inner iterations and the maximum physical time must be defined.

For Iterations 2–4 presented here, a time step of $1 \times 10^{-4}$ was used with 20 inner iterations per time step. However, a numerical error caused Iteration 3 to prematurely stop at 0.63 s of the full 1.3 s start-up time. Iteration 4 used the available data from Iteration 3 to define the outlet pressure for 0 – 0.62490 s, and supplemented the missing data with data from Iteration 2 for 0.62495 – 1.31 s. The time step was kept at $1 \times 10^{-4}$ for Iteration 4, which ran with no numerical errors. The time step was then reduced to $1 \times 10^{-5}$ for Iteration 5 in an effort to improve solution accuracy. The
Figure 5.4: Flow chart describing simulation iteration process
time step of $1 \times 10^{-4}$ corresponds to $0.8^\circ$ of rotation per time step at the highest rotational speed, while the smaller time step of $1 \times 10^{-5}$ corresponds to $0.08^\circ$ of rotation per time step at the highest rotational speed. Prior work centered around similar situations has shown that a time step resulting in $0.5^\circ$ of rotation per time step is sufficiently small for good simulation convergence [16].

The maximum physical time is defined by the start-up rotation curve. The full start-up can be seen in Fig. 5.3, and defines maximum physical time for all simulations to be 1.31 s.

5.5 Results

The main feature of the method of iterating through simulations to approach a constant inlet total pressure presented here is adjusting the pressure outlet boundary condition based on the inlet total pressure data of the previous simulation iteration. Figure 5.5 shows the normalized difference between inlet total pressure and target inlet total pressure during start-up for each simulation iteration. Iteration 1 is a single phase constant outlet pressure simulation. Iterations 2 – 5 are cavitating simulations that use data from previous iterations to define outlet static pressure.

As seen in Fig.5.5, and as should be expected, the inlet total pressure difference varies wildly from the target inlet total pressure in Iteration 1. Inlet total pressure is initially much higher than the target, but drops rapidly, and falls below the target from 0 - 0.2 seconds. Inlet total pressure then increases back up to approximately the starting value from 0.2 to 1.3 s. The average difference between actual and target inlet total pressure for Iteration 1 during start-up is 118%. As head coefficient and rotational rate are directly related, and as outlet pressure for Iteration 1 is constant through start-up as seen in Fig. 5.6, inlet total pressure changes drastically to accommodate the transient rotational rate.

The inlet total pressure trend of Iteration 2 is significantly different than that of Iteration 1 due to the changing outlet pressure based on the inlet total pressure of Iteration 1. The inlet total pressure difference between actual and target of Iteration 2 is essentially constant and negligible between 0 and 0.08 s. Pressure difference increases the difference between actual and target inlet total pressure to 49% on average between 0.08 and 0.25 s after which it returns back to approximately 0 at 0.7 s, where it remains until it oscillates rapidly from 1.2 to 1.3 s. The average difference between actual and target inlet total pressure for Iteration 2 during the full start-up is 2.59%. As can be seen in Fig. 5.6, the defined pressure on the outlet boundary condition varies
Figure 5.5: Normalized difference between actual inlet total pressure and target inlet total pressure during start-up for each simulation iteration

Figure 5.6: Normalized outlet static pressure during start-up for each simulation iteration
with time to counteract the inlet total pressure changes in Iteration 1. For Iteration 2, outlet pressure increases sharply from 0 to 0.23 s, and decreases from 0.23 to 1.3 s.

Inlet total pressure difference more closely approximates a constant value with no difference between target and actual pressure during the full start-up with Iteration 3. The deviation from target pressure from 0.2 to 0.7 s seen in Iteration 2 decreases by up to 75% in Iteration 3 which results in an average difference of 1.29% from 0 to 0.63 s. However, as can be seen in Fig. 5.5, the inlet total pressure data for Iteration 3 ends at 0.63 s. This simulation iteration experienced a numerical error at this point, preventing simulation completion. The incomplete outlet pressure data of Iteration 3 is seen in Fig. 5.6. Outlet pressure corresponding to Iteration 3 exhibits an average decrease of 3.6% when compared to Iteration 2 from 0 to 0.63 s. Despite the incomplete status of this simulation, the inlet pressure data from 0 to 0.63 s provides useful information that was used to modify the outlet pressure definition for Iteration 4.

As Iteration 3 stopped prematurely, solution data from Iteration 2 was used to define outlet pressure for Iteration 4 from 0.63 – 1.31 s. As seen in Fig. 5.5, the inlet pressure difference for Iteration 4 is slightly better than Iteration 3, with an average reduction in inlet total pressure of 2.4% between 0 and 0.4 s. However, from 0.5 to 0.6 s, there is a ~ 4% average increase in inlet pressure difference of Iteration 4 over Iteration 3. After 0.6 s, the inlet pressure difference for Iteration 4 closely follows that of Iteration 2, with the same oscillation from 1.2 to 1.3 s. The average difference between inlet total pressure and target inlet total pressure for Iteration 4 during the full start-up is 1.56%. As seen in Fig. 5.6, the outlet pressure of Iteration 4 closely follows, but is slightly lower than that of Iteration 3, corresponding to a 3.6% decrease, on average, from 0 to 0.63 s. From 0.63 to 1.3 s, the outlet pressure of Iteration 4 closely follows that of Iteration 2, with an average decrease of 4.8%.

Using the data from Iteration 4 to generate a new commanded outlet pressure trace, shown in Fig. 5.6, results in the curve corresponding to Iteration 5, which has an average decrease of 21% when compared to that of Iteration 4. The improved outlet pressure definition causes an average inlet pressure difference difference of 1.27% for Iteration 5 during startup. Using the method of controlling simulation inlet total pressure during start-up described in the methods section above, average difference during start-up dropped from 118% to 1.27% in five simulation iterations.
Figure 5.7: Head coefficient produced by the impeller during start-up for each simulation iteration

Figure 5.7 plots pump head coefficient through the start-up process. Each simulation iteration predicts approximately the same result, with only a 7% decrease on average between Iteration 1 and Iteration 5 from 0 to 1 s. From 1 to 1.3 s, head coefficient exhibits strong oscillations due to pressure fluctuations caused by cavity formation and collapse. Head coefficient is a non-dimensional measure of the pump performance, and describes the difference between inlet and outlet boundaries, which is why it varies so little between Iteration 1 and Iteration 5.

Cavity formation is shown in Figure 5.10, which plots total cavity volume in the computational domain during start-up. In all simulation iterations, cavity volume rapidly increases from 0.08 to 0.22 s, after which it decreases to approximately zero cavity volume at 0.65 s. Rapid cavity volume increases correspond directly to the region of greatest rotational acceleration in the start-up curve. The vapor collapse at 0.65 s causes inlet pressure fluctuations that are evident in Fig. 5.5, and increase the difficulty of maintaining a constant inlet total pressure. The formation of another cavity occurs at 0.86 s and dissipates at 1.244 s. This cavity is different from that occurring from 0.08 to 0.22 s, in that it forms on the pressure side of the blade and at the blade tips rather than on the typical suction side. Figure 5.8 shows an isosurface of 30% vapor fraction forming on the pressure side of the blades at ~ 0.98 s of Iteration 5. Figure 5.9 provides another view of the pressure side cavitation formation at the same time in Iteration 5, and provides a contour plot of vapor
fraction on a cross section of the computational domain. Both cavity volume nodes exist in all cavitating simulations, but cavity volume increases with increasing simulation iteration. Iteration 4 and 5 agree within 5% of the cavity volume between 0.08 and 0.22 s. However, Iteration 5 predicts cavity volume between 0.86 and 1.22 s to be up to 79% greater but on average 13% greater than Iteration 4.

The second node of cavitation formation from 0.8 to 1.2 s is a function of fluctuating flow coefficient affecting incidence and causing flow separation. Figure 5.11 plots flow coefficient through the pump during start-up. Flow coefficient varies between Iteration 1 and 2, but seems to converge on a solution for Iterations 3, 4, and 5, with an average decrease of 1.8% from Iteration 4 to 5. For these simulations, flow coefficient increases rapidly from 0 to 0.4 s, which is when the impeller reaches the maximum rotational velocity. From 0.4 to 0.8, flow coefficient continues to increase, but at a lesser rate as the impeller is decelerating. From 0.8 to 1.3 s, flow coefficient begins to increase again, at a 95% greater slope than that from 0.4 to 0.8 s. The increasing flow through the pump must cause the flow angle at the leading edge to increase so much such that it causes flow separation on the pressure side of the impeller blades.

Figure 5.12 plots the normalized RMS force on the impeller during start-up. Force magnitudes vary widely from Iteration 1 to Iteration 5, but Iterations 3, 4, and 5 seem to agree well, suggesting that a reasonably converged force solution was reached. However, average difference
Figure 5.9: Contour plot of vapor fraction on a cross section of the domain. This provides another view of the cavitation formation on the pressure side of the impeller blades in Iteration 5.

Figure 5.10: Normalized cavity volume during start-up for each simulation iteration
between the RMS forces of Iterations 4 and 5 is 87%. This is a function of the increased cavitation in Iteration 5 causing force fluctuations, which increases the average difference. In Iteration 5, RMS force initially drops from 0 to 0.13 s. An increase in force occurs from 0.13 to 0.28, which corresponds to the rapid formation of cavitation. Forces then decrease to approximately 0 at 0.48 s and remain there until 0.64 s. Large force fluctuations, similar to those seen in the inlet and outlet pressure plots (Figs. 5.5 and 5.6), are evident at 0.64 s due to cavitation collapse. Forces then increase to a maximum until 0.95 s, which is maintained at an approximately constant level until 1.3 s. The fluctuations from 1 to 1.3 s are caused by cavitation volumes forming on the pressure side of the blades, which loads and unloads the impeller.

5.6 Conclusions

This work described a novel method to simulate the transient start-up of a centrifugal impeller using CFD. This method consists of using a non cavitating simulation to generate a typical inlet pressure data trace for a pump. The difference between the inlet total pressure data and the target inlet total pressure is then used to adjust the defined static pressure on the outlet boundary condition in an effort to maintain a constant inlet total pressure in a cavitating simulation. Inlet total pressure is not constant during the first iteration of the cavitating simulation, so a second it-
Figure 5.12: Normalized RMS force on the impeller during start-up for each simulation iteration

eration of the cavitating simulation is completed using the difference between inlet total pressure of the first cavitating simulation and the target inlet total pressure. This difference is again used to adjust the defined outlet static pressure. The process of iterating through simulations modified using inlet total pressure difference data from the previous simulation was completed 4 times for this work. Reasonable agreement was achieved between simulation Iterations 4 and 5, with inlet total pressure varying by $\sim 8\%$ on average, head coefficient varying by $\sim 11\%$ on average, cavitation volume varying by $\sim 3\%$ on average, flow coefficient varying by $\sim 2\%$ on average, and RMS forces varying by $\sim 3\%$ on average. The relatively low difference between critical values of Iterations 4 and 5 suggests that a reasonable solution was achieved in 5 total simulation iterations.

Cavitation formation and collapse dominates the flow physics of this pump during start-up. During the initial rapid rotational acceleration of the pump, large cavity volumes form, but rapidly collapse as rotational speed stabilizes for a short time. This rapid formation and collapse of vapor induces pressure fluctuations that strongly affects the pressure field and also imparts large rotordynamic forces to the impeller. In addition to the primary cavitation formation when the pump experiences the greatest rotational acceleration, a secondary cavity forms on the pressure side of the impeller blades as a result of incidence changes. Corresponding pressure fluctuations
and a large increase in rotordynamic forces accompany the formation and collapse of the secondary cavity.

While this simulation method seems to accurately model the complex flow during a transient start up, additional work could be done to further improve and verify the data presented here and validate the method. The commanded outlet pressure curves could be smoothed to potentially limit pressure and force fluctuations which could remove the potential of error propagating through multiple simulation iterations and improve solution accuracy. Additionally, experimental data of a pump during fast start up must be obtained and compared to simulation data for the same pump. This would provide an absolute measure of the accuracy of the simulation method set forth here.
CHAPTER 6. CONCLUSIONS

Various aspects of inducer performance have been investigated with computational fluid dynamics simulations. Multiple inducer geometries operating at multiple flow conditions were simulated. Simulations of a 7° inducer with and without tip leakage flow were performed at on-and off-design flow coefficients with and without cavitation. In cases where backflow is present, two vortex structures upstream of the leading edge are evident: a small tip vortex and a larger backflow region. Removing the tip leakage flow was shown to increase backflow upstream penetration by up to 14%. At all flow coefficients below the design point, the inducer that experienced tip leakage flow exhibited less backflow upstream penetration. This suggests that tip leakage flow has a negligible effect in backflow formation. The tip leakage flow was found to instead contribute to the strength of a vortex located at the blade tips just upstream of the leading edge of the inducer blades. Backflow upstream propagation was found to always increase with increasing inlet diffusion in both single phase and under cavitating conditions where the effective flow area in the throat is reduced. The strong relationship between backflow and inlet diffusion indicates the primary mechanism for backflow is high inducer inlet diffusion due to the high flow incidence at the leading edge.

Simulations exploring the thermal effects of cavitation on the performance of a 9.4° inducer with and without an SCD were also performed. As liquid rocket turbopumps frequently use liquid hydrogen as the working fluid, these simulations allow for a more detailed investigation of inducer cavitating performance than simulations using cold water as the working fluid. These simulations showed that no significant head improvement resulting from SCD implementation was apparent at flow coefficients near the design point. SCD implementation was found to increase inducer stability by maintaining low incidence values at all simulated flow coefficients. Low incidence values caused by SCD implementation were also found to limit backflow formation and upstream propagation. In liquid hydrogen, the inducer with no SCD produced up to 43% greater head
than the inducer with an SCD at NPSH values corresponding to cavitation breakdown. When breakdown is defined as a 3\% reduction in head coefficient from the non cavitating value, the predicted breakdown NPSH was slightly lower for the inducer with an SCD operating at $\phi = 0.117, 0.107,$ and $0.08$. While breakdown NPSH improvements were evident with the simulated $9.4^\circ$ inducer and an SCD in liquid hydrogen, they are not ubiquitous, and will vary with inducer and SCD geometries. It is only at low flow coefficients ($\sim 60\%$ of design) that the SCD appeared to provide any head improvements when pumping liquid hydrogen. At $\sim 60\%$ of the design flow coefficient, the SCD provided an increase in head coefficient of $4.4\%$ over the inducer with no SCD. Additionally, at $\sim 60\%$ of the design flow point, the flow field of the inducer with no SCD was unstable enough such that simulations were unable to reach a converged solution. At this same flow point the solutions for the inducer with the SCD remained stable enough to continue simulating the flow deep into breakdown.

While the SCD provided little to no real benefit in head coefficient at the simulated flow points, significant improvements in rotor stability were noted. Increased stability is a function of low incidence over the range of simulated flow coefficients caused by increased mass flow at the blade leading edge from the SCD reinjection slots. The mass flow increase supplied by the SCD was shown to increase linearly with inlet diffusion for this SCD geometry in liquid hydrogen, allowing for a greater reduction in incidence at lower flow coefficients where incidence naturally increases due to decreasing flow angles. The low incidence maintained by the SCD at flow coefficients far below design also limits flow diffusion and stall at the blade inlets. The limited flow diffusion in turn resulted in reduced backflow upstream penetration. All the benefits of SCD implementation in liquid hydrogen were a result of limiting incidence at low flow coefficients.

At all flow coefficients below $\phi = 0.117$, the SCD significantly repressed backflow formation which can provide myriad benefits in a pump, including a reduction in cavitation instabilities due to a stabilized upstream flow field. The reduction in backflow formation decreases structural forces on the inducer at all flow coefficients below $\phi = 0.117$, and was more apparent at flow coefficients far below design. SCD implementation was found to allow an inducer to operate stably without backflow over a wider range of flow coefficients than is possible with no SCD, even with liquid hydrogen as the working fluid.
Another aspect of inducer performance explored here centered around cavitation formation and growth during the fast start-up and shut-down of a centrifugal impeller. A novel method to simulate the transient start-up of a centrifugal impeller using CFD was developed and used to predict inducer cavitation formation and performance. This simulation method consisted of using a non cavitating simulation to generate a typical inlet pressure data trace for a pump. The difference between the inlet total pressure data and the target inlet total pressure was used to adjust the defined static pressure on the outlet boundary condition in an effort to maintain a constant inlet total pressure in the cavitating simulation. Inlet total pressure was not constant during the first iteration of the cavitating simulation, and a second iteration of the cavitating simulation was completed using the difference between inlet total pressure of the first cavitating simulation and the target inlet total pressure. The difference was again used to adjust the defined outlet static pressure. The process of iterating through simulations modified using inlet total pressure difference data from the previous simulation was completed 5 times. Reasonable agreement was achieved between simulation Iterations 4 and 5, with inlet total pressure varying by $\sim 8\%$ on average, head coefficient varying by $\sim 11\%$ on average, cavitation volume varying by $\sim 3\%$ on average, flow coefficient varying by $\sim 2\%$ on average, and RMS forces varying by $\sim 3\%$ on average. The relatively low difference between critical values of Iterations 4 and 5 suggests that a reasonable solution was achieved in 5 total simulation iterations.

Cavitation formation and collapse dominated the flow physics of this pump during start-up. During the initial rapid rotational acceleration of the pump, large cavity volumes formed, but rapidly collapsed as rotational speed stabilized for a short time. This rapid formation and collapse of vapor induced pressure fluctuations that strongly affected the pressure field and imparted large rotordynamic forces to the impeller. In addition to the primary cavitation formation when the pump experiences the greatest rotational acceleration, a secondary cavity formed on the pressure side of the impeller blades due to of incidence changes induced by pump deceleration. Corresponding pressure fluctuations and a large increase in rotordynamic forces accompanied the formation and collapse of the secondary cavity.
6.1 Recommendations for Further Investigation

The most important area for further investigation centers around improving inducer design with SCD design in liquid hydrogen. The SCD design considered in Chapter 4 was added to a well-established inducer design and resulted in little to no suction performance improvements at relatively common flow points. Additional improvements to suction performance at flow coefficients near the design point will likely result from an inducer designed in tandem with an SCD. As SCD implementation maintains low incidence and improves inducer stability, traditional inducer design parameters may not have the same effect when implemented with SCD design. Additional simulations exploring inducer and SCD design should be performed, while also considering the thermal effects that suppress cavitation at cryogenic temperatures.

Additional work to improve the simulation of liquid rocket turbomachines pumping liquid hydrogen would also be beneficial. The cryogenic simulations performed for this work employed a steady solver that did not capture transient cavitation growth or transient rotodynamic forces. Time-accurate simulations would provide important transient data to aid in the design and function of new inducer/SCD pairs.

While the simulation method to model the transient start-up of a centrifugal impeller seemed to accurately model the complex flow during a transient start up, additional work should be done to further improve and verify the data presented here and validate the method. The commanded outlet pressure curves could be smoothed to potentially limit pressure and force fluctuations which could remove the potential of error propagating through multiple simulation iterations and improve solution accuracy. To properly validate the accuracy of the method, experimental data is needed. Experimental data of a pump during fast start up should be obtained and compared to simulation data for the same pump. This would provide an absolute measure of the accuracy of the simulation method set forth here. The potential benefits of this simulation method could be valuable for future pump design. But a lack of method validation significantly weakens the current viability.
REFERENCES


128


APPENDIX A. CRUNCH SIMULATION TUTORIAL

This appendix contains a tutorial that develops a simulation setup of a liquid hydrogen axial flow inducer with no SCD. This is the same process to used to generate the simulation results presented in Chapter 4. As CRUNCH is most often run using a command line, it can sometimes be confusing and unintuitive compared to running simulations in Star-CCM+. This section is meant to provide a broad overview of how the simulations presented in Chapter 4 were set up and run. For additional details and tutorials, refer to the CRUNCH documentation [72, 79]. The setup and commands included here were developed for the supercomputing resources made available by the BYU Office of Research Computing, and may vary depending on application. A Unix/Linux tutorial is provided by the BYU Office of Research Computing, and provides useful background information [97].

A.1 Simulation Steps

The steps involved in running CRUNCH simulations to generate a breakdown curve are: 1) Pre-process mesh using PRECRUNCH, 2) Set up module specific and global input files for a single phase case, 3) Run the CRUNCH_INCOMP module and obtain a single phase solution, 4) Postprocess the single phase solution for plot and diagnostic files, 5) Set up module specific input file for a cavitating case, 6) Run the CRUNCH_INCOMP module starting from the single phase solution to obtain a cavitating solution, 7) Postprocess the cavitating solution for plot and diagnostic files, 8) Modify module specific input file to run at a lower cavitation number, 9) Repeat steps 6-8 to generate a breakdown curve.

A.2 Mesh Generation

As with most CFD, the first step to running a CRUNCH simulation is the mesh. CRUNCH has no built-in mesh generation capabilities, and all meshes must be generated externally and
imported into CRUNCH. The CRUNCH meshes used for this work were generated in Autogrid by Jamin Bitter at Concepts NREC. These meshes consist of two files: a grid file (casename.g) and a boundary condition definition file (casename.bcs). The grid and boundary condition definition files work together to define the mesh, and both must be located in the same directory where the simulation will be run. This directory will be referred to as the working directory for the remainder of this appendix.

A.3 Mesh Preprocessing

Before the simulation can be run, the mesh files must be preprocessed to decompose the mesh into load balanced domains for parallel computations. This is done with the utility PRECRUNCH, which is included in the CRUNCH installation package. While PRECRUNCH can be run in interactive or batch mode, PRECRUNCH must be run interactively when processing a new mesh. Running PRECRUNCH will generate the proper structure in the working directory. This is done with the following commands, entered into the command line, where the -i flag calls PRECRUNCH interactively. Changing the -i flag to -f causes PRECRUNCH to run in batch mode.

module load crunch_cfd/3.0
PRECRUNCH.exe -i casename

The user is then prompted for information about the mesh and simulation setup. For the simulations completed for this work, the grid generator is Autogrid as previously stated. Additionally, there are five viscous boundaries: each inducer blade, the inducer hub, and the shroud. The user must then input the boundary tags corresponding to these surfaces. This can be found by loading the mesh into the CRUNCH GUI and visually determining the boundary tags, or by convention. The meshes made for this work always correspond the boundary tag 31 with the inducer hub, 32 with the inducer shroud, and 300 – 302 with the inducer blades.

The user is then asked if grid quality diagnostics should be performed. This process has no affect on the solution other than computational time, and the additional mesh quality information can useful in troubleshooting any mesh issues. When grid quality diagnostics are performed, an additional file in the working directory (casename_gridquality.dat) is generated. Grid quality diagnostics were performed for all simulations in this work.
The user is then asked if the solution files should be partitioned. This option only makes sense if a solution restart file exists in the working directory. As PRECRUNCH was only used for this work to prepare a new mesh, no restart solutions were available, and the default no was input each time. It should be noted that the solution partitioning process can be completed by itself if a solution file exists in the working directory and PRECRUNCH is run in interactive mode. This is useful when returning to complete a solution or using a solution as initial conditions for another simulation.

The user is then prompted to input if periodic boundaries exist. This follows the same process as the viscous boundary definition. However, no periodic boundaries exist in any of the meshes used for this work, so this setting was always set to the default no.

The user will be prompted for mesh partitioning strategies. All simulations for this work used the Recursive coordinate bisection method with 256 partitions. The number of partitions must match the $NUM\_PROCS$ flag in the module input file and the $module\_nprocs$ flag in the global input file. The output type should be set to the default of unformatted, and no batch mode input file should be written. Autogrid generates a formatted file, and the user should deviate from the default and indicate the file is formatted when prompted in the Autogrid to PRECRUNCH conversion. Once this is complete, PRECRUNCH will run and generate the following directories in the working directory: GRID, MVGRD, OUTPUT, SOLUTION, UNSTEADY. The two directories of note are the OUTPUT and SOLUTION directories. The OUTPUT directory contains the output from each processor, which can be analyzed during troubleshooting if the simulation fails prematurely. The SOLUTION directory contains the partitioned solution files, which are combined later into a single file with POSTCRUNCH.

A.4 Single Phase Module Input File Setup

At this point, a module input file is required, as no batch input file was written in PRECRUNCH. A general structure can be built using the CRUNCH GUI, but should be checked by hand. The CRUNCH GUI was not used for simulation generation for this work, so this appendix makes no attempt to describe GUI usage. Refer to the CRUNCH documentation for additional GUI usage information [72]. Section A.5 is a module input file that should be named casename.inp and includes brief comments explaining each flag to simulate a three bladed inducer rotating at 30000
rpm at a flow coefficient of $\phi = 0.107$, an inlet temperature of 21.1 K, and an outlet pressure of 400 kPa. This input file does not activate the cavitation model, and is used to simulate single-phase performance and generate initial conditions for a cavitating simulation. A complete listing of the input file flags is available in the CRUNCH user manual [72], but a select few will be discussed here.

The &BCS section of the input file defines the boundary conditions. Flow coefficient is controlled by changing the $USUBIN(1)$ flag, which defines inlet velocity in $m/s$. Inlet static temperature in K is defined with the $TSUBIN(1)$ flag. Cavitation number or NPSH is defined with the $PBACK(1)$ flag, which defines outlet static pressure. The $FIX\_BACKFLOW(1)$ and $FIX\_OUTFL\_VTAN(1)$ flags are disabled in the simulations presented here. $FIX\_BACKFLOW(1)$ allows outlet pressure to vary in the event of reverse flow. $FIX\_OUTFL\_VTAN(1)$ specifies a radial pressure gradient based on the average swirl velocity at the outlet boundary. This is comparable to the radial pressure outlet definition of Star-CCM+ described in Chapter 2. $FIX\_OUTFL\_VTAN(1)$ was disabled for the simulations presented here, but could be used for future simulations.

As mentioned earlier, the shroud corresponds to BCTAG 32. This boundary is defined to rotate at 30000 RPM with the flag $OMEGAX\_SURF(2)$. The direction of rotation is dictated by the mesh coordinate system, and the right-hand rule. In this case, a negative sign is used to rotate the shroud CCW, to simulate the inducer rotating CW along the positive x-axis. The user should be reminded that rotation is only possible around the x-axis in CRUNCH. Rotation is handled the same way with BCTAGS 300, 301 and 302, which correspond to each inducer blade. No other surface has an active $OMEGAX\_SURF$ flag.

The $INTERNAL\_BC\_TYPE$ flags define surfaces to be internal interfaces. Two matching surfaces are required to generate the interface. The $INTERNAL\_BC\_KIND$ flag is set to 0 for closest node with the $INTERNAL\_BC\_AVERAGE\_TYPE$ set to $-1$ for no averaging as the mesh is fully annular and contains no periodic surfaces.

With the boundary conditions fully defined, the remaining parameters must be addressed. $ITEMP$ is set to 1 in $&PRIMARY\_FLUID\_SELECTIONS$ and $&REFERENCE$ to model a cryogenic fluid. $ICAVITATION$ is initially set to 0 in $&PRIMARY\_FLUID\_SELECTIONS$ and $&CAVITATION$ to run a single phase simulation. $IROTATION$ is sent to 1 in $&PRIMARY\_MODEL\_SELECTIONS$ and $&ROTATION$ with $ROTX\_SPEED$ set to 30000 RPM. The flag $CFL\_MAX$ in $&SCHEME$ is
initially set to 0.5. This specifies the Courant-Friedrichs-Lewy (CFL) number, and is applied to the pseudo-time step. The CFL number is set low to start the simulation to aid in convergence and prevent errors that could stop the simulation. As the simulation progresses, this number is increased.

A.5 Single Phase Module Input File

! Single Phase Flow Coefficient 0.107 Temperature 21.1 K

&PRIMARY_FLUID SELECTIONS

ITEMP = 1, !Choose ideal (constant property) or cryogenic fluid (properties change with temperature). Cryogenic fluid requires temperature field solution.

ICAVITATION = 0, !Select cavitation flow model

SPECIFY_CAVNUM = 0, !Choose whether vapor will be entered directly or calculated from cavitation number in CAVITATION namelist.

ISURF = 0, !Solve for bubble surface area

/

&PRIMARY_MODEL_SELECTIONS

ITIME = 0, !Steady-state or time-accurate calculation, and time derivative order of accuracy for unsteady simulations.

IVISC = 1, !Specify calculation to be viscous

ITURB = 2, !Specify whether or not k-epsilon turbulence modeling is activated.

UNST_TURB = 0, !Select unsteady turbulence model (none, hybrid RANS–LES, or LES model). Not an actual input,
used to set HYBRID and ILES in the TURBULENCE namelist.

IROTATION = 1, !Activate domain rotation
MOVING_GRID = 0, !Activate grid movement
ADAPT_GRID = 0, !Activate CRISP CFD.
/

&PRIMARY_SOLVER_SELECTIONS
RESTART_SOLUTION = 1, !Specify whether or not the solution shall be restarted from a previous run.
NMAX = 15000, !Specify number of time steps to be run.
/

&PROPERTIES
RHOGAS = 1, !Specify constant vapor phase density (kg/m^3) for an ideal fluid.
RHOLIQ = 1000, !Specify constant liquid phase density (kg /m^3) for an ideal fluid.
CRYO_FLUID_TYPE = 1, !Specify cryogenic fluid type.
ZPRL = 1.228, !Specify liquid Prandtl number to get liquid conductivity from viscosity.
ZPRV = 0.775, !Specify vapor Prandtl number to get liquid conductivity from viscosity.
/

&REFERENCE
CYL_CORD = true, !Specify flow solution coordinate frame.
Essential for periodic BC in theta—direction.
ITEMP = 1, !Choose ideal or cryogenic fluid. Properties of cryogenic fluids change with temperature and requires solution of the temperature field.
ISURF = 0, !Specify whether or not the bubble radius/surface area is solved for.
PREFD = 200000, !Specify Reference Free Stream Pressure (Pa)
QINFD = 10.3493, !Reference velocity is used to non-dimensionalize velocity components, enthalpy, and computer Reynolds number
TINFD = 21.1, !Freestream reference temperature for computing fluid properties
IVISC = 1, !Specify calculation to be viscous
RESTART_PSHIFT = 0,

/

&TURBULENCE
USE_KE_VALUE = true, !Indicate how eddy viscosity is specified.
ITURB = 2, !Specify whether or not k-epsilon turbulence modeling is activated.
WALL_FUNCTION = true, !Specify if wall function procedure or near-wall damping procedure is to be used
IKEW = 3, !Specify near-wall damping model.
PERATIO = 100, !Specify a numerical cap on the production to dissipation rate ratio
ZMUMAX = 25000, !Specify a numerical cap for the turbulent viscosity

/
&ROTATION

IROTATION = 1, ! Specify if domain rotation is activated.
ROTX_SPEED = 30000, ! Rotation speed about the x–axis in RPM (positive clockwise, negative counterclockwise).
/

&CAVITATION

ICAVITATION = 0, ! Select the cavitation flow model.
CAV_BACK = 1500, ! Specify backward cavitation rate.
CAV_FWD = 750, ! Specify forward cavitation rate.
CAVIT_FIX = true, ! Makes the vapor volume fraction eqn first order (useful for low grid resolution)
/

&SCHHEME

BETA_MAX = 2500, ! Parameter that scales the acoustic speed to get well conditioned eigenvalues.
FIRST_ORDER = false, ! Specify order of accuracy for spatial integration.
L2RECON = false, ! Specify gradient reconstruction procedure used to generate higher order flux.
EPC = 0.1, ! Specify an entropy fix for acoustic eigenvalues in the upwind flux calculation.
EPU = 0.1, ! Specify an entropy fix for scalar eigenvalues in the upwind flux calculation.
PHIGCAP = 0.5, ! Fix to prevent overshoot and undershoot of vapor volume fraction.
IMODE = 0, ! Specify time–stepping scheme.
ITIME = 0, ! Select the order of accuracy of time derivative for unsteady simulations
CFL_MAX = 4.0, !Specify CFL number applied to the pseudo-
time step.
CFL_PHYS = 1, !Specify CFL number applied to the physical
-time step.
IMPL_FLOW_SOLV = true, !Choose implicit or explicit
solution integration.
/

&BCS

SUBIN_BC_TYPE(1) = 1, !Select Boundary Condition Type
SUBTYPE(1) = 1,
USUBIN(1) = 10.3493,
VSUBIN(1) = 0,
WSUBIN(1) = 0,
INFLAG(1) = 0, !Switch to outflow if flow going out of
domain. Back pressure given by Static Pressure imposed
. 0: Pure Inflow Boundary
PHIGSUBIN(1) = 0, !Cannot be zero if bubbly model is used
.
TSUBIN(1) = 21.1, !Inflow static Temperature.
XKSUBIN(1) = 0, !Specify k in m^2/s^2. Code computes if
not specified.
XESUBIN(1) = 0, !Specify epsilon in m^2/s^3. Code
computes if not specified.
SUBIN_PROFILE(1) = .F., !Subsonic inflow read from file
BCNAME(1) = 'inlet', !Specify a name for the boundary
condition.
SUBOUT_BC_TYPE(1) = 2, !Select Boundary Condition Type
PBACK(1) = 400000, !Back pressure (Pa).
SUBOUT_AMPL(1) = 0, !Non-dimensional amplitude of PBACK for fluctuating outflow condition.
SUBOUT_FREQ(1) = 0, !Frequency (in Hz) of PBACK for fluctuating outflow condition.
FIX BACKFLOW(1) = false, !A correction to allow the back pressure to float if reverse flow occurs. This fix is applied if the local velocity at the outflow boundary falls below a prescribed cut-off velocity. F: No Correction
FIX OUTFL_VTAN(1) = false, !Choose whether to manually specify a swirl velocity, or have the code automatically compute radial pressure gradient at the outflow. T: Specify Swirl Velocity
VTANFIX(1) = 0, !Used to prescribe the swirl velocity (m/s) at the outflow boundary that will be used to calculate the radial pressure gradient.
VARY PBACK(1) = false, !Vary the back pressure to achieve a target mass flow rate. F: Fixed Back Pressure
BCNAME(2) = 'outlet', !Specify a name for the boundary condition.
WALL BC_TYPE(1) = 30, !Select Boundary Condition Type
BCNAME(30) = 'bctag030', !Specify a name for the boundary condition.
NOSLIP BC_TYPE(1) = 31, !Select Boundary Condition Type
ADIABATIC WALL(1) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall
USURF(1) = 0, !Wall u-velocity (for moving walls) for the viscous surface (m/s)
VSURF(1) = 0, !Wall v-velocity (for moving walls) for the viscous surface (m/s)
WSURF(1) = 0, !Wall w-velocity (for moving walls) for the viscous surface (m/s)

OMEGAX_SURF(1) = 0, !X-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAY_SURF(1) = 0, !Y-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAZ_SURF(1) = 0, !Z-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

XC_SURF(1) = 0, !X-component of center of rotation for rotating domains (grid units).

YC_SURF(1) = 0, !Y-component of center of rotation for rotating domains (grid units).

ZC_SURF(1) = 0, !Z-component of center of rotation for rotating domains (grid units).

BCNAME(31) = 'hub', !Specify a name for the boundary condition.

NOSLIP_BC_TYPE(2) = 32, !Select Boundary Condition Type

ADIABATIC_WALL(2) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall

USURF(2) = 0, !Wall u-velocity (for moving walls) for the viscous surface (m/s)

VSURF(2) = 0, !Wall v-velocity (for moving walls) for the viscous surface (m/s)

WSURF(2) = 0, !Wall w-velocity (for moving walls) for the viscous surface (m/s)
OMEGAX\_SURF(2) = -30000, \( !X \)-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAY\_SURF(2) = 0, \( !Y \)-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAZ\_SURF(2) = 0, \( !Z \)-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

XC\_SURF(2) = 0, \( !X \)-component of center of rotation for rotating domains (grid units).

YC\_SURF(2) = 0, \( !Y \)-component of center of rotation for rotating domains (grid units).

ZC\_SURF(2) = 0, \( !Z \)-component of center of rotation for rotating domains (grid units).

BCNAME(32) = 'shroud', ! Specify a name for the boundary condition.

INTERNAL\_BC\_TYPE(1) = 70, ! Select Boundary Condition Type

INTERNAL\_BC\_PAIR(1) = 71, ! Specify which internal boundary is the partner boundary.

INTERNAL\_BC\_KIND(1) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node

INTERNAL\_BC\_AVERAGE\_TYPE(1) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging

BCNAME(70) = 'bctag070', ! Specify a name for the boundary condition.

INTERNAL\_BC\_TYPE(2) = 71, ! Select Boundary Condition Type

INTERNAL\_BC\_PAIR(2) = 70, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(2) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(2) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(71) = 'bctag071', ! Specify a name for the boundary condition.
INTERNAL_BC_TYPE(3) = 72, ! Select Boundary Condition Type
INTERNAL_BC_PAIR(3) = 73, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(3) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(3) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(72) = 'bctag072', ! Specify a name for the boundary condition.
INTERNAL_BC_TYPE(4) = 73, ! Select Boundary Condition Type
INTERNAL_BC_PAIR(4) = 72, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(4) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(4) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(73) = 'bctag073', ! Specify a name for the boundary condition.
INTERNAL_BC_TYPE(5) = 74, ! Select Boundary Condition Type
INTERNAL_BC_PAIR(5) = 75, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(5) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(5) = -1, !Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(74) = 'bctag074', !Specify a name for the boundary condition.
INTERNAL_BC_TYPE(6) = 75, !Select Boundary Condition Type
INTERNAL_BC_PAIR(6) = 74, !Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(6) = 0, !Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(6) = -1, !Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(75) = 'bctag075', !Specify a name for the boundary condition.
NOSLIP_BC_TYPE(3) = 300, !Select Boundary Condition Type
ADIABATIC_WALL(3) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall
USURF(3) = 0, !Wall u-velocity (for moving walls) for the viscous surface (m/s)
VSURF(3) = 0, !Wall v-velocity (for moving walls) for the viscous surface (m/s)
WSURF(3) = 0, !Wall w-velocity (for moving walls) for the viscous surface (m/s)
OMEGAX_SURF(3) = 0, !X-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAY_SURF(3) = 0, !Y-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAZ_SURF(3) = 0, \( \text{!Z--component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.} \)

XC_SURF(3) = 0, \( \text{!X--component of center of rotation for rotating domains (grid units).} \)

YC_SURF(3) = 0, \( \text{!Y--component of center of rotation for rotating domains (grid units).} \)

ZC_SURF(3) = 0, \( \text{!Z--component of center of rotation for rotating domains (grid units).} \)

BCNAME(300) = 'blad1', \( \text{!Specify a name for the boundary condition.} \)

NOSLIP_BC_TYPE(4) = 301, \( \text{!Select Boundary Condition Type} \)

ADIABATIC_WALL(4) = true, \( \text{!Adiabatic or isothermal wall condition. T: Adiabatic Wall} \)

USURF(4) = 0, \( \text{!Wall u--velocity (for moving walls) for the viscous surface (m/s)} \)

VSURF(4) = 0, \( \text{!Wall v--velocity (for moving walls) for the viscous surface (m/s)} \)

WSURF(4) = 0, \( \text{!Wall w--velocity (for moving walls) for the viscous surface (m/s)} \)

OMEGAX_SURF(4) = 0, \( \text{!X--component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.} \)

OMEGAY_SURF(4) = 0, \( \text{!Y--component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.} \)

OMEGAZ_SURF(4) = 0, \( \text{!Z--component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.} \)
XC_SURF(4) = 0, !X–component of center of rotation for rotating domains (grid units).
YC_SURF(4) = 0, !Y–component of center of rotation for rotating domains (grid units).
ZC_SURF(4) = 0, !Z–component of center of rotation for rotating domains (grid units).

BCNAME(301) = 'blade2', !Specify a name for the boundary condition.

NOSLIP_BC_TYPE(5) = 302, !Select Boundary Condition Type
ADIABATIC_WALL(5) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall

USURF(5) = 0, !Wall u–velocity (for moving walls) for the viscous surface (m/s)
VSURF(5) = 0, !Wall v–velocity (for moving walls) for the viscous surface (m/s)
WSURF(5) = 0, !Wall w–velocity (for moving walls) for the viscous surface (m/s)

OMEGAX_SURF(5) = 0, !X–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAY_SURF(5) = 0, !Y–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAZ_SURF(5) = 0, !Z–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

XC_SURF(5) = 0, !X–component of center of rotation for rotating domains (grid units).
YC_SURF(5) = 0, !Y–component of center of rotation for rotating domains (grid units).
ZC_SURF(5) = 0, !Z–component of center of rotation for rotating domains (grid units).

BCNAME(302) = 'blade3', !Specify a name for the boundary condition.

/&PROPZONES

INIT_ZONE = 1, !Specify number of initial property zones.
ZONE_GRID_TYPE(1) = 2, !Select zone type Cylindrical Shell

XMIN_ZONE(1) = −0.01, !Specify zonal minimum x coordinate in grid units.
XMAX_ZONE(1) = 0.08, !Specify zonal maximum x coordinate in grid units.
YMIN_ZONE(1) = −0.5, !Specify box zone minimum y coordinate in grid units.
YMAX_ZONE(1) = 0.56, !Specify box zone maximum y coordinate in grid units.
ZMIN_ZONE(1) = −0.5, !Specify box zone minimum z coordinate in grid units.
ZMAX_ZONE(1) = 0.56, !Specify box zone maximum z coordinate in grid units.
RADMIN_ZONE(1) = 0, !Specify minimum radius in grid units.
RADMAX_ZONE(1) = 0.1, !Specify maximum radius in grid units.
XC_ZONE(1) = 0.06, !Specify x coordinate in grid units.
YC_ZONE(1) = 0, !Specify y coordinate in grid units.
ZC_ZONE(1) = 0, !Specify z coordinate in grid units.
ZONE_VEL_TYPE(1) = 2, ! Select zone velocity coordinate system. Cylindrical
U_ZONE(1) = 0.113219, ! Enter velocity component along the x-axis (m/s).
VR_ZONE(1) = 0, ! Enter radial velocity component (m/s).
VT_ZONE(1) = 0, ! Enter tangential velocity component (m/s).
P_ZONE(1) = 100000, ! Enter the initial zone pressure.
PHIG_ZONE(1) = 0, ! Enter the vapor–phase volume fraction.
T_ZONE(1) = 21.1, ! Enter the initial zone temperature in K.
XK_ZONE(1) = 0, ! Specify k in m^2/s^2. Code will compute if not specified.
XE_ZONE(1) = 0, ! Specify epsilon in m^2/s^3. Code will compute if not specified.

&FIXES
/

&IO
NMAX = 15000, ! Specify number of time steps to be run.
RESTART = false, ! Specify whether current calculation will restart from a previous solution.
STORE_FREQ = 100, ! Specify frequency for intermediate restart files to be written out during a run.
IPLOT = 99999, ! Specify frequency for intermediate plot file output.
&TRACES
/

&FORCEM

IFORCEM = 4, ! 0: No force and moment calculation, N>0: calculate force and moment for N surface tags.

TAG_PSURF(1) = 31, ! Specify BC tag number of the surface for which the forces and moments are computed.

TAG_PSURF_TYPE(1) = 1, ! Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC

XC_PSURF(1) = 0, ! Specify x–center of moment location.
YC_PSURF(1) = 0, ! Specify y–center of moment location.
ZC_PSURF(1) = 0, ! Specify z–center of moment location.

AREA_PSURF(1) = 1, ! Specify reference area for non–dimensionalizing forces.

XLEN_PSURF(1) = 1, ! Specify reference length for non–dimensionalizing moments (with AREA_PSURF).

TAG_PSURF(2) = 300, ! Specify BC tag number of the surface for which the forces and moments are computed.

TAG_PSURF_TYPE(2) = 1, ! Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC

XC_PSURF(2) = 0, ! Specify x–center of moment location.
YC_PSURF(2) = 0, ! Specify y–center of moment location.
ZC_PSURF(2) = 0, ! Specify z–center of moment location.

AREA_PSURF(2) = 1, ! Specify reference area for non–dimensionalizing forces.

XLEN_PSURF(2) = 1, ! Specify reference length for non–dimensionalizing moments (with AREA_PSURF).

TAG_PSURF(3) = 301, ! Specify BC tag number of the surface for which the forces and moments are computed.
TAG_Psurf_Type(3) = 1, !Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC
XC_Psurf(3) = 0, !Specify x−center of moment location.
YC_Psurf(3) = 0, !Specify y−center of moment location.
ZC_Psurf(3) = 0, !Specify z−center of moment location.
AREA_Psurf(3) = 1, !Specify reference area for non−dimensionalizing forces.
XLEN_Psurf(3) = 1, !Specify reference length for non−dimensionalizing moments (with AREA_Psurf).

TAG_Psurf(4) = 302, !Specify BC tag number of the surface for which the forces and moments are computed.
TAG_Psurf_Type(4) = 1, !Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC
XC_Psurf(4) = 0, !Specify x−center of moment location.
YC_Psurf(4) = 0, !Specify y−center of moment location.
ZC_Psurf(4) = 0, !Specify z−center of moment location.
AREA_Psurf(4) = 1, !Specify reference area for non−dimensionalizing forces.
XLEN_Psurf(4) = 1, !Specify reference length for non−dimensionalizing moments (with AREA_Psurf).

&DIAGBCS

DIAGBC_FREQ = 25, !Frequency for performing the BC diagnostics.
DIAGBC_DIM = true, !Specify dimensional or non−dimensional output.
DIAGBC_LIST = 1,2, !DIAGBC_LISTSet the array of diagnostic BC tags.

/
&VOLUME_INTEGRATION

PERFORM_VOLUME_INTEGRATION = .T., !Activates the volume
integration diagnostic routines.
N_BOX_VOLUME_INTEGRATION = 1, !Specify number of regions
in which to perform the volume integration.
XMIN_VOLUME_INTEGRATION(1) = −0.56, !Specify minimum x−
coordinate.
XMAX_VOLUME_INTEGRATION(1) = 0.59, !Specify maximum x−
coordinate.
YMIN_VOLUME_INTEGRATION(1) = −0.56, !Specify minimum y−
coordinate.
YMAX_VOLUME_INTEGRATION(1) = 0.56, !Specify maximum y−
coordinate.
ZMIN_VOLUME_INTEGRATION(1) = −0.56, !Specify minimum z−
coordinate.
ZMAX_VOLUME_INTEGRATION(1) = 0.56, !Specify maximum z−
coordinate.
/

&ACTUATOR
/

&PRECRUNCH

CASENAME = 'casename',
BINREAD = .T., !The read−in (casename.precrunch) is
binary.
BINWRITE = .T., !The restart file is binary.
RESTART = .T., !PRECRUNCH initialized by PRECRUNCH file
!Viscous Boundary Condition Setup

154
VISCOUS = .T., !Viscous boundaries present
VISCBCS = 31,32,300,301,302, !sum
NVISCBC = 5, !sum sum
VISCD = .T., !Calculate wall distances for viscous grids.
!Mesh–Grid Partitioning Properties
IPARTTYPE = 1, !Specify mesh partitioning strategy
NUM_PROCS = 256, !Number of processors to use
!Periodic Boundary Condition Setup
IPRDTYPE = 4, !Specify type of periodicity the boundary represents.
!Option to partition existing solution (.rso CRUNCH restart file).
SOLUTION = .T., !Partition solution restart file into Solution directory
SOLNFORM = 'unformatted', !Specify read-in solution restart file format
!Activate Grid Quality Metrics
GRIDGEN = F,
GGHYBRID = F,
VGRIDNS = F,
COBALT = F,
FLUENT = F,
SOLIDMESH = F,
ICEMGRID = F,
AUTOGRID = F,
NUMWALLS = 5,
IWALL_BC_TYPE = 31,
MVGRD = F,
MZONE = F,
NMAP = F,
EXTRUSION = F,
PLOTFILE = F,
RETAG_PCFILE = "",
PERIODIC_PAIR_TOLERANCE = 9.999997473787516E−005,
/

&POSTCRUNCH

! CFD File Format Specification
BINARY = true, ! Specify how CRUNCH CFD files were written.

! Create Plot Files
RECOMBINE_PLOT = .T., ! Recombine CRUNCH plot files from
partitioned files.
PLOT_FORMAT = 4, ! Specify the plot file format.
NONDIM = .F., ! Write non-dimensional plot variables.
! Recombine solution file from partitioned data.
RECOMBINE_RST = .T., ! Recombine CRUNCH CFD solution from
partitioned files.
RSNBNIN = true, ! Specify recombined CRUNCH solution file
format.
! Transfer a previous solution to a new grid.
SOLN_TRANSFER = .F., ! Transfer a previous solution to a
new grid. Requires either a recombined existing old
solution file, or a SOLUTION subdirectory populated
with casename.rsnNNN file(s).
SOLN_INTERP_ORDER = 0, ! Choose cell containment for grids
where surface distance smaller than grid spacing.
RSNFILE = 'casename.rsn', ! Specify the name of the
original recombined solution.
NEWBIN = true, !Specify file format for newly interpolated solution file.

!Create new restart file with additional species.
SOLN_ADDSPECIES = .F., !Add more species to an existing solution file.
RSNFILE_ADDSPECIES = casename.rsn', !Specify name of source solution file to augment.
NEWCASE_ADDSPECIES = 'new_casename', !Specify casename of the target (augmented) solution file.
nSpOld = , !Specify number of chemical species in source solution.
cSpeciesNamesOld = , !Specify source species list and order.

!Process transient trace data.
PROCESS_TRACE = .F., !Post-process trace point transient data.
TRACE_EXTRACT_OPTION = 1, !Specify how to write out extracted trace data.
TRACEFILE = 'trace.dat', !Specify the filename of the tracepoint file.

!Recombine time-averaged (QBar) files.
RECOMBINE_QBAR = .F., !Recombine CRUNCH solution from partitioned files (QBar).
QBARBIN = true, !Specify CRUNCH time-averaged qbar file format.
QBARFORM = 1, !Specify the output format of the recombinated time-averaged qbar file for plotting with Ensight or FieldView.

!Generate solution difference report.
SOLN_DIFFERENCE = .F., !Create a solution difference report. Valid only if both old and new restart file(s) exist.

!Extract solution data from BC surfaces.
SURF_WRITE_DATA = .F., !Extract data from a BC surface to a specified output file.
SURF_BCTAG = 1, !Specify surface BC tag.
SURF_NUM_VARS = 1, !Specify the number of variables to extract.
SURF_VARLIST(1) = 1, !Select variables Fluid Density
SURF_FILENAME = 'casename_surface.data', !Specify the filename of the surface data file.
SURF_APPEND = .F., !Append extracted data to the existing surface file.
/

A.6 Single Phase Global Input File Setup

Section A.7 provides the global input file for this simulation. This file is simple, and just needs to match the module_nprocs and module_case flags with the corresponding flags in the module input file.

A.7 Single Phase Global Input File

namelist / global / module_name, module_case, module_nprocs
&GLOBAL
  module_name = 'CRUNCH_INCOMP',
  module_nprocs = 256,
  module_case = 'casename',
/

158
&GUI_CONFIG

    GUI_VERSION = '2.6.2_r.0',
    VERBOSE_LABELS = 1,
    ADVANCED_OPTIONS = 0,
    VISUALIZER = 1,
    COMMENTS = 1,

/

&RUNTIME

    PROFILE = 'Local',
    REMOTE_PATH = '',
    STATUS_FREQ = 10,
    CHECKUP_FREQ = 15,
    CHECKUP_RETRY = 5,

/

A.8 Run Single Phase Simulation

Section A.9 provides a file used to schedule and submit the simulation on the BYU supercomputer. The file should be named and saved as submit_job. This file schedules a CRUNCH job to run for a maximum of 72 hours on 256 processor cores, each using 4 GB of memory on 16 nodes. This file also defines the job to run with infiniband on the rhel7 resources, which are relevant at the time of writing. The resources and corresponding definitions may change in the future. After the job completes, it will send the user a notification email. For more information about these settings, visit the BYU Office of Research Computing website [98]. The submit file runs CRUNCH multiple times, while varying parameters to reach a converged solution quickly without needing to schedule a new job. The submit file calls the global and module input files to run a simulation. After the simulation has completed, it also calls the file trans.py, included in the CRUNCH installation package to transfer the solution after a simulation has run. This file then
calls POSTCRUNCH to prepare the simulation data for an external postprocessing software package. POSTCRUNCH is run in batch mode, which is defined by the flags in the module input file section &POSTCRUNCH. The input file provided in Section A.9 is set up to provide output files using SI units (NONDIM = .F. flag) in a file that can be read by ParaView (PLOT_FORMAT = 4 flag). After POSTCRUNCH completes, the submit file uses the GUI input editor, which is a tool available in the CRUNCH installation package to increase the CFL number and number of iterations to complete. This process is completed multiple times, each time increasing the CFL number and number of iterations to run to efficiently reach a converged simulation solution. This submit file may or may not result in a converged solution, and may need to be used again with further increased CFL numbers. This will need to be determined on a case-by-case basis. The job is submitted with the following command entered into the command line:

```
sbatch submit_job
```

### A.9 Supercomputer Run File

```
#!/bin/bash

# Submit this script with: sbatch submit_job

#SBATCH --time=72:00:00    # walltime
#SBATCH --ntasks=256       # number of processor cores (i.e. tasks)
#SBATCH --mem-per-cpu=4G   # memory per CPU core
#SBATCH --licenses=crunch_mphys:1
#SBATCH --nodes=16          # number of nodes
#SBATCH --C 'ib&rhel7'     # features syntax (use quotes): –C ’a&b&c &d’
#SBATCH --mail-user= xxxx@xxxx.com  # include relevant email address
```

160
#SBATCH −−mail−type=BEGIN
#SBATCH −−mail−type=END
#SBATCH −−mail−type=FAIL

# Compatibility variables for PBS. Delete if not needed.
export PBS_NODEFILE=‘/fslapps/fslutils/generate_pbs_nodefile’
export PBS_JOBID=$SLURM_JOB_ID
export PBS_O_WORKDIR="$SLURM_SUBMIT_DIR”
export PBS_QUEUE=batch

cd $SLURM_SUBMIT_DIR
printf ”Job debugging info

Node List”
cat $PBS_NODEFILE
printf “

Current Directory: $PWD

Starting CRUNCher run at ”
date
printf “

module purge
module load gcc/8
module load openmpi/3.1
module load crunch_cfd/3.0

srun CRUNCHCFD.py
trans.py
POSTCRUNCH.exe −f casename
GUI_INPUTEDITOR.exe -b RESTART CRUNCH_INCOMP true
GUI_INPUTEDITOR.exe -b CFL_MAX CRUNCH_INCOMP 0.5
GUI_INPUTEDITOR.exe -b NMAX CRUNCH_INCOMP 1000
srun CRUNCHCFD.py
trans.py
POSTCRUNCH.exe -f casename

GUI_INPUTEDITOR.exe -b CFL_MAX CRUNCH_INCOMP 0.75
GUI_INPUTEDITOR.exe -b NMAX CRUNCH_INCOMP 2000
srun CRUNCHCFD.py
trans.py
POSTCRUNCH.exe -f casename

GUI_INPUTEDITOR.exe -b CFL_MAX CRUNCH_INCOMP 1
GUI_INPUTEDITOR.exe -b NMAX CRUNCH_INCOMP 5000
srun CRUNCHCFD.py
trans.py
POSTCRUNCH.exe -f casename

GUI_INPUTEDITOR.exe -b CFL_MAX CRUNCH_INCOMP 1.5
srun CRUNCHCFD.py
trans.py
POSTCRUNCH.exe -f casename

GUI_INPUTEDITOR.exe -b CFL_MAX CRUNCH_INCOMP 2
srun CRUNCHCFD.py
trans.py
POSTCRUNCH.exe -f casename

printf "-----\n"

162
printf "Ending CRUNCHer run at 

date
printf "\n---\n
exit 0

A.10  Post Process Single Phase Simulation

At this point, a solution has been reached and POSTCRUNCH has generated Paraview solution files. The casename.vtu file is the solution for the full computational domain and can be loaded directly into Paraview for further analysis. The other .vtu files are solution files corresponding to specific surfaces, and can also be loaded directly into Paraview. The files casename_flow.data001 and casename_flow.data002 provide tabular data at the inlet and outlet boundary conditions for each iteration. The files casename_forcem.data031, casename_forcem.data300, casename_forcem.data301, casename_forcem.data302, and casename_forcem_global.data provide force information on the rotor hub, rotor blades and the global forces, respectively. These require processing using Matlab or something similar. The processing method of these data files is left to the user.

A.11  Cavitating Module Input File Setup

Once a single phase solution has converged, a cavitating simulation can be run. Section A.12 includes the module input file modified to model cavitation. The only difference occurs in the &PRIMARY_FLUID_SELECTIONS and &CAVITATION sections, where the flag is changed by the user for to ICAVITATION = 1 to turn on the cavitation model for a new simulation. Initially, the outlet pressure is set to the same pressure as the single phase outlet pressure to minimize differences to improve simulation convergence. Outlet pressure is then marched down in a series of simulations to generate a breakdown curve. Each step down in outlet pressure should be started from the converged solution of the previous simulation.
A.12 Cavitating Module Input File

! Cavitating Flow Coefficient 0.107 Temperature 21.1 K

&PRIMARY_FLUID_SELECTIONS

ITEMP = 1, !Choose ideal (constant property) or cryogenic fluid (properties change with temperature). Cryogenic fluid requires temperature field solution.
ICAVITATION = 1, !Select cavitation flow model
SPECIFY_CAVNUM = 0, !Choose whether vapor will be entered directly or calculated from cavitation number in CAVITATION namelist.
ISURF = 0, !Solve for bubble surface area
/

&PRIMARY_MODEL_SELECTIONS

ITIME = 0, !Steady-state or time-accurate calculation, and time derivative order of accuracy for unsteady simulations.
IVISC = 1, !Specify calculation to be viscous
ITURB = 2, !Specify whether or not k-epsilon turbulence modeling is activated.
UNST_TURB = 0, !Select unsteady turbulence model (none, hybrid RANS–LES, or LES model). Not an actual input, used to set HYBRID and ILES in the TURBULENCE namelist.
IROTATION = 1, !Activate domain rotation
MOVING_GRID = 0, !Activate grid movement
ADAPT_GRID = 0, !Activate CRISP CFD.
/

164
&PRIMARY_SOLVER_SELECTIONS

RESTART_SOLUTION = 2, !Specify whether or not the solution shall be restarted from a previous run.
NMAX = 5000, !Specify number of time steps to be run.
/

&PROPERTIES

RHOGAS = 1, !Specify constant vapor phase density (kg/m^3) for an ideal fluid.
RHOLIQ = 1000, !Specify constant liquid phase density (kg/m^3) for an ideal fluid.
CRYO_FLUID_TYPE = 1, !Specify cryogenic fluid type.
ZPRL = 1.228, !Specify liquid Prandtl number to get liquid conductivity from viscosity.
ZPRV = 0.775, !Specify vapor Prandtl number to get liquid conductivity from viscosity.
/

&REFERENCE

CYL_CORD = true, !Specify flow solution coordinate frame.
  Essential for periodic BC in theta—direction.
ITEMP = 1, !Choose ideal or cryogenic fluid. Properties of cryogenic fluids change with temperature and requires solution of the temperature field.
ISURF = 0, !Specify whether or not the bubble radius/surface area is solved for.
PREFD = 200000, !Specify Reference Free Stream Pressure (Pa)

165
QINFD = 10.3493, !Reference velocity is used to non-dimensionalize velocity components, enthalpy, and computer Reynolds number
TINFD = 21.1, !Freestream reference temperature for computing fluid properties
IVISC = 1, !Specify calculation to be viscous
RESTART_PSHIFT = 0,
/
&TURBULENCE
USE_KE_VALUE = true, !Indicate how eddy viscosity is specified.
ITURB = 2, !Specify whether or not k-epsilon turbulence modeling is activated.
WALL_FUNCTION = true, !Specify if wall function procedure or near-wall damping procedure is to be used
IKEW = 3, !Specify near-wall damping model.
PERATIO = 100, !Specify a numerical cap on the production to dissipation rate ratio
ZMUMAX = 25000, !Specify a numerical cap for the turbulent viscosity
/
&ROTATION
IROTATION = 1, !Specify if domain rotation is activated.
ROTX_SPEED = 30000, !Rotation speed about the x-axis in RPM (positive clockwise, negative counter-clockwise).
/
&CAVITATION

166
ICAVITATION = 1, ! Select the cavitation flow model.
CAV_BACK = 1500, ! Specify backward cavitation rate.
CAV_FWD = 750, ! Specify forward cavitation rate.
CAVIT_FIX = true, ! Makes the vapor volume fraction eqn first order (useful for low grid resolution)
/

&SCHEME

BETA_MAX = 2500, ! Parameter that scales the acoustic speed to get well conditioned eigenvalues.
FIRST_ORDER = false, ! Specify order of accuracy for spatial integration.
L2RECON = false, ! Specify gradient reconstruction procedure used to generate higher order flux.
EPC = 0.1, ! Specify an entropy fix for acoustic eigenvalues in the upwind flux calculation.
EPU = 0.1, ! Specify an entropy fix for scalar eigenvalues in the upwind flux calculation.
PHIGCAP = 0.5, ! Fix to prevent overshoot and undershoot of vapor volume fraction.
IMODE = 0, ! Specify time-stepping scheme.
ITIME = 0, ! Select the order of accuracy of time derivative for unsteady simulations
CFL_MAX = 2.0, ! Specify CFL number applied to the pseudo-time step.
CFL_PHYS = 1, ! Specify CFL number applied to the physical-time step.
IMPL_FLOW_SOLV = true, ! Choose implicit or explicit solution integration.
/

167
SUBINBC_TYPE(1) = 1, !Select Boundary Condition Type
SUBTYPE(1) = 1,
USUBIN(1) = 10.3493,
VSUBIN(1) = 0,
WSUBIN(1) = 0,
INFLAG(1) = 0, !Switch to outflow if flow going out of domain. Back pressure given by Static Pressure imposed.
0: Pure Inflow Boundary
PHIGSUBIN(1) = 0, !Cannot be zero if bubbly model is used.
TSUBIN(1) = 21.1, !Inflow static Temperature.
XKSUBIN(1) = 0, !Specify k in m^2/s^2. Code computes if not specified.
XESUBIN(1) = 0, !Specify epsilon in m^2/s^3. Code computes if not specified.
SUBIN_PROFILE(1) = .F., !Subsonic inflow read from file
BCNAME(1) = 'inlet', !Specify a name for the boundary condition.
SUBOUTBC_TYPE(1) = 2, !Select Boundary Condition Type
PBACK(1) = 400000, !Back pressure (Pa).
SUBOUTAMPL(1) = 0, !Non-dimensional amplitude of PBACK for fluctuating outflow condition.
SUBOUTFREQ(1) = 0, !Frequency (in Hz) of PBACK for fluctuating outflow condition.
FIX_BACKFLOW(1) = false, !A correction to allow the back pressure to float if reverse flow occurs. This fix is applied if the local velocity at the outflow boundary
falls below a prescribed cut-off velocity. F: No Correction

FIX_OUTFL_VTAN(1) = false, !Choose whether to manually specify a swirl velocity, or have the code automatically compute radial pressure gradient at the outflow. T: Specify Swirl Velocity

VTANFIX(1) = 0, !Used to prescribe the swirl velocity (m/s) at the outflow boundary that will be used to calculate the radial pressure gradient.

VARY_PBACK(1) = false, !Vary the back pressure to achieve a target mass flow rate. F: Fixed Back Pressure

BCNAME(2) = 'outlet', !Specify a name for the boundary condition.

WALL_BC_TYPE(1) = 30, !Select Boundary Condition Type

BCNAME(30) = 'bctag030', !Specify a name for the boundary condition.

NOSLIP_BC_TYPE(1) = 31, !Select Boundary Condition Type

ADIABATIC_WALL(1) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall

USURF(1) = 0, !Wall u-velocity (for moving walls) for the viscous surface (m/s)

VSURF(1) = 0, !Wall v-velocity (for moving walls) for the viscous surface (m/s)

WSURF(1) = 0, !Wall w-velocity (for moving walls) for the viscous surface (m/s)

OMEGAX_SURF(1) = 0, !X-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAY_SURF(1) = 0, !Y–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAZ_SURF(1) = 0, !Z–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
XC_SURF(1) = 0, !X–component of center of rotation for rotating domains (grid units).
YC_SURF(1) = 0, !Y–component of center of rotation for rotating domains (grid units).
ZC_SURF(1) = 0, !Z–component of center of rotation for rotating domains (grid units).
BCNAME(31) = 'hub', !Specify a name for the boundary condition.
NOSLIP_BC_TYPE(2) = 32, !Select Boundary Condition Type
ADIABATIC_WALL(2) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall
USURF(2) = 0, !Wall u–velocity (for moving walls) for the viscous surface (m/s)
VSURF(2) = 0, !Wall v–velocity (for moving walls) for the viscous surface (m/s)
WSURF(2) = 0, !Wall w–velocity (for moving walls) for the viscous surface (m/s)
OMEGAX_SURF(2) = −30000, !X–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAY_SURF(2) = 0, !Y–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAZ_SURF(2) = 0, !Z-component of angular velocity for
rotating walls (rpm). This is relative to the
rotational frame of reference.

XC_SURF(2) = 0, !X-component of center of rotation for
rotating domains (grid units).

YC_SURF(2) = 0, !Y-component of center of rotation for
rotating domains (grid units).

ZC_SURF(2) = 0, !Z-component of center of rotation for
rotating domains (grid units).

BCNAME(32) = 'shroud', !Specify a name for the boundary
condition.

INTERNAL_BC_TYPE(1) = 70, !Select Boundary Condition Type
INTERNAL_BCPAIR(1) = 71, !Specify which internal
boundary is the partner boundary.

INTERNAL_BC_KIND(1) = 0, !Select periodicity type for the
internal boundaries. 0: Closest Node

INTERNAL_BC_AVERAGE_TYPE(1) = -1, !Select averaging type
for the internal boundaries. -1: No Averaging

BCNAME(70) = 'bctag070', !Specify a name for the boundary
condition.

INTERNAL_BC_TYPE(2) = 71, !Select Boundary Condition Type
INTERNAL_BCPAIR(2) = 70, !Specify which internal
boundary is the partner boundary.

INTERNAL_BC_KIND(2) = 0, !Select periodicity type for the
internal boundaries. 0: Closest Node

INTERNAL_BC_AVERAGE_TYPE(2) = -1, !Select averaging type
for the internal boundaries. -1: No Averaging

BCNAME(71) = 'bctag071', !Specify a name for the boundary
condition.

INTERNAL_BC_TYPE(3) = 72, !Select Boundary Condition Type
INTERNAL_BC_PAIR(3) = 73, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(3) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(3) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(72) = ’bctag072’, ! Specify a name for the boundary condition.
INTERNAL_BC_TYPE(4) = 73, ! Select Boundary Condition Type
INTERNAL_BC_PAIR(4) = 72, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(4) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(4) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(73) = ’bctag073’, ! Specify a name for the boundary condition.
INTERNAL_BC_TYPE(5) = 74, ! Select Boundary Condition Type
INTERNAL_BC_PAIR(5) = 75, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(5) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node
INTERNAL_BC_AVERAGE_TYPE(5) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging
BCNAME(74) = ’bctag074’, ! Specify a name for the boundary condition.
INTERNAL_BC_TYPE(6) = 75, ! Select Boundary Condition Type
INTERNAL_BC_PAIR(6) = 74, ! Specify which internal boundary is the partner boundary.
INTERNAL_BC_KIND(6) = 0, ! Select periodicity type for the internal boundaries. 0: Closest Node

INTERNAL_BC_AVERAGE_TYPE(6) = -1, ! Select averaging type for the internal boundaries. -1: No Averaging

BCNAME(75) = 'bctag075', ! Specify a name for the boundary condition.

NOSLIP_BC_TYPE(3) = 300, ! Select Boundary Condition Type
ADIABATIC_WALL(3) = true, ! Adiabatic or isothermal wall condition. T: Adiabatic Wall

USURF(3) = 0, ! Wall u-velocity (for moving walls) for the viscous surface (m/s)
VSURF(3) = 0, ! Wall v-velocity (for moving walls) for the viscous surface (m/s)
WSURF(3) = 0, ! Wall w-velocity (for moving walls) for the viscous surface (m/s)

OMEGAX_SURF(3) = 0, ! X-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAY_SURF(3) = 0, ! Y-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAZ_SURF(3) = 0, ! Z-component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

XC_SURF(3) = 0, ! X-component of center of rotation for rotating domains (grid units).

YC_SURF(3) = 0, ! Y-component of center of rotation for rotating domains (grid units).

ZC_SURF(3) = 0, ! Z-component of center of rotation for rotating domains (grid units).
BCNAME(300) = 'blade1', !Specify a name for the boundary condition.
NOSLIP_BC_TYPE(4) = 301, !Select Boundary Condition Type
ADIABATIC_WALL(4) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall
USURF(4) = 0, !Wall u–velocity (for moving walls) for the viscous surface (m/s)
VSURF(4) = 0, !Wall v–velocity (for moving walls) for the viscous surface (m/s)
WSURF(4) = 0, !Wall w–velocity (for moving walls) for the viscous surface (m/s)
OMEGAX_SURF(4) = 0, !X–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAY_SURF(4) = 0, !Y–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
OMEGAZ_SURF(4) = 0, !Z–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.
XC_SURF(4) = 0, !X–component of center of rotation for rotating domains (grid units).
YC_SURF(4) = 0, !Y–component of center of rotation for rotating domains (grid units).
ZC_SURF(4) = 0, !Z–component of center of rotation for rotating domains (grid units).
BCNAME(301) = 'blade2', !Specify a name for the boundary condition.
NOSLIP_BC_TYPE(5) = 302, !Select Boundary Condition Type
ADIABATIC_WALL(5) = true, !Adiabatic or isothermal wall condition. T: Adiabatic Wall

USURF(5) = 0, !Wall u–velocity (for moving walls) for the viscous surface (m/s)

VSURF(5) = 0, !Wall v–velocity (for moving walls) for the viscous surface (m/s)

WSURF(5) = 0, !Wall w–velocity (for moving walls) for the viscous surface (m/s)

OMEGAX_SURF(5) = 0, !X–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAY_SURF(5) = 0, !Y–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

OMEGAZ_SURF(5) = 0, !Z–component of angular velocity for rotating walls (rpm). This is relative to the rotational frame of reference.

XC_SURF(5) = 0, !X–component of center of rotation for rotating domains (grid units).

YC_SURF(5) = 0, !Y–component of center of rotation for rotating domains (grid units).

ZC_SURF(5) = 0, !Z–component of center of rotation for rotating domains (grid units).

BCNAME(302) = 'blade3', !Specify a name for the boundary condition.

/ &PROPZONES

INIT_ZONE = 1, !Specify number of initial property zones.
ZONE_GRID_TYPE(1) = 2, !Select zone type Cylindrical Shell
XMIN ZONE(1) = −0.01, !Specify zonal minimum x coordinate in grid units.
XMAX ZONE(1) = 0.08, !Specify zonal maximum x coordinate in grid units.
YMIN ZONE(1) = −0.5, !Specify box zone minimum y coordinate in grid units.
YMAX ZONE(1) = 0.56, !Specify box zone maximum y coordinate in grid units.
ZMIN ZONE(1) = −0.5, !Specify box zone minimum z coordinate in grid units.
ZMAX ZONE(1) = 0.56, !Specify box zone maximum z coordinate in grid units.
RADMIN ZONE(1) = 0, !Specify minimum radius in grid units.
RADMAX ZONE(1) = 0.1, !Specify maximum radius in grid units.
XC ZONE(1) = 0.06, !Specify x coordinate in grid units.
YC ZONE(1) = 0, !Specify y coordinate in grid units.
ZC ZONE(1) = 0, !Specify z coordinate in grid units.
ZONE_VEL_TYPE(1) = 2, !Select zone velocity coordinate system. Cylindrical
U ZONE(1) = 0.113219, !Enter velocity component along the x–axis (m/s).
VR ZONE(1) = 0, !Enter radial velocity component (m/s).
VT ZONE(1) = 0, !Enter tangential velocity component (m/s).
P ZONE(1) = 100000, !Enter the initial zone pressure.
PHIG ZONE(1) = 0, !Enter the vapor–phase volume fraction.
T_ZONE(1) = 21.1, !Enter the initial zone temperature in K.

XK_ZONE(1) = 0, !Specify k in m^2/s^2. Code will compute if not specified.

XE_ZONE(1) = 0, !Specify epsilon in m^2/s^3. Code will compute if not specified.

&FIXES

&IO

NMAX = 5000, !Specify number of time steps to be run.

RESTART = true, !Specify whether current calculation will restart from a previous solution.

STORE_FREQ = 100, !Specify frequency for intermediate restart files to be written out during a run.

IPLOT = 99999, !Specify frequency for intermediate plot file output.

&TRACES

&FORCEM

IFORCEM = 4, !0: No force and moment calculation, N>0: calculate force and moment for N surface tags.

TAG_Psurf(1) = 31, !Specify BC tag number of the surface for which the forces and moments are computed.
TAG PSURF Type(1) = 1, ! Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC
XC PSURF(1) = 0, ! Specify x–center of moment location.
YC PSURF(1) = 0, ! Specify y–center of moment location.
ZC PSURF(1) = 0, ! Specify z–center of moment location.
AREA PSURF(1) = 1, ! Specify reference area for non–dimensionalizing forces.

XLEN PSURF(1) = 1, ! Specify reference length for non–dimensionalizing moments (with AREA PSURF).

TAG PSURF(2) = 300, ! Specify BC tag number of the surface for which the forces and moments are computed.
TAG PSURF Type(2) = 1, ! Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC
XC PSURF(2) = 0, ! Specify x–center of moment location.
YC PSURF(2) = 0, ! Specify y–center of moment location.
ZC PSURF(2) = 0, ! Specify z–center of moment location.
AREA PSURF(2) = 1, ! Specify reference area for non–dimensionalizing forces.

XLEN PSURF(2) = 1, ! Specify reference length for non–dimensionalizing moments (with AREA PSURF).

TAG PSURF(3) = 301, ! Specify BC tag number of the surface for which the forces and moments are computed.
TAG PSURF Type(3) = 1, ! Specify BC type (wall or inflow) for which the forces and moments are computed. Wall BC
XC PSURF(3) = 0, ! Specify x–center of moment location.
YC PSURF(3) = 0, ! Specify y–center of moment location.
ZC PSURF(3) = 0, ! Specify z–center of moment location.
AREA PSURF(3) = 1, ! Specify reference area for non–dimensionalizing forces.
XLEN_PSURF(3) = 1, !Specify reference length for non-
dimensionalizing moments (with AREA_PSURF).
TAG_PSURF(4) = 302, !Specify BC tag number of the surface
for which the forces and moments are computed.
TAG_PSURF_TYPE(4) = 1, !Specify BC type (wall or inflow)
for which the forces and moments are computed. Wall BC
XC_PSURF(4) = 0, !Specify x−center of moment location.
YC_PSURF(4) = 0, !Specify y−center of moment location.
ZC_PSURF(4) = 0, !Specify z−center of moment location.
AREA_PSURF(4) = 1, !Specify reference area for non−
dimensionalizing forces.
XLEN_PSURF(4) = 1, !Specify reference length for non−
dimensionalizing moments (with AREA_PSURF).

/ &DIAGBCS

DIAGBC_FREQ = 25, !Frequency for performing the BC
diagnostics.
DIAGBC_DIM = true, !Specify dimensional or non−
dimensional output.
DIAGBCLIST = 1,2, !DIAGBCLISTSet the array of diagnostic
BC tags.

/ &VOLUME_INTEGRATION

PERFORM_VOLUME_INTEGRATION = .T., !Activates the volume
integration diagnostic routines.
N_BOX_VOLUME_INTEGRATION = 1, !Specify number of regions
in which to perform the volume integration.
XMIN_VOLUME_INTEGRATION(1) = -0.56, !Specify minimum x-coordinate.
XMAX_VOLUME_INTEGRATION(1) = 0.59, !Specify maximum x-coordinate.
YMIN_VOLUME_INTEGRATION(1) = -0.56, !Specify minimum y-coordinate.
YMAX_VOLUME_INTEGRATION(1) = 0.56, !Specify maximum y-coordinate.
ZMIN_VOLUME_INTEGRATION(1) = -0.56, !Specify minimum z-coordinate.
ZMAX_VOLUME_INTEGRATION(1) = 0.56, !Specify maximum z-coordinate.

/ 

&ACTUATOR

/ 

&PRECRUNCH

CASENAME = 'casename',
BINREAD = .T., !The read-in (casename.precrunch) is binary.
BINWRITE = .T., !The restart file is binary.
RESTART = .T., !PRECRUNCH initialized by PRECRUNCH file
!Viscous Boundary Condition Setup
VISCOUS = .T., !Viscous boundaries present
VISCBCS = 31,32,300,301,302, !sum
NVISCBC = 5, !sum sum
VISC = .T., !Calculate wall distances for viscous grids.
!Mesh–Grid Partitioning Properties
IPARTTYPE = 1, !Specify mesh partitioning strategy
NUM_PROCS = 256, ! Number of processors to use

! Periodic Boundary Condition Setup
IPRDTYPE = 4, ! Specify type of periodicity the boundary represents.

! Option to partition existing solution (.rso CRUNCH restart file).
SOLUTION = .T., ! Partition solution restart file into Solution directory
SOLNFORM = 'unformatted', ! Specify read-in solution restart file format

! Activate Grid Quality Metrics
GRIDGEN = F,
GGHYBRID = F,
VGRIDNS = F,
COBALT = F,
FLUENT = F,
SOLIDMESH = F,
ICEMGRID = F,
AUTOGRID = F,
NUMWALLS = 5,
I_WALL_BC_TYPE = 31,
MGRID = F,
MZONE = F,
NMAP = F,
EXTRUSION = F,
PLOTFILE = F,
RETAG_PCFILE = "",
PERIODIC_PAIR_TOLERANCE = 9.9999997473787516E-005,
/

181
&POSTCRUNCH

!CFD File Format Specification

BINARY = true, ! Specify how CRUNCH CFD files were written.

! Create Plot Files

RECOMBINE_PLOT = .T., ! Recombine CRUNCH plot files from partitioned files.

PLOT_FORMAT = 4, ! Specify the plot file format.

NONDIM = .F., ! Write non-dimensional plot variables.

! Recombine solution file from partitioned data.

RECOMBINE_RST = .T., ! Recombine CRUNCH CFD solution from partitioned files.

RSN_BIN = true, ! Specify recombined CRUNCH solution file format.

! Transfer a previous solution to a new grid.

SOLN_TRANSFER = .F., ! Transfer a previous solution to a new grid. Requires either a recombined existing old solution file, or a SOLUTION subdirectory populated with casename.rsnNNN file(s).

SOLN_INTERP_ORDER = 0, ! Choose cell containment for grids where surface distance smaller than grid spacing.

RSNFILE = 'casename.rsn', ! Specify the name of the original recombined solution.

NEWBIN = true, ! Specify file format for newly interpolated solution file.

! Create new restart file with additional species.

SOLN_ADDSPECIES = .F., ! Add more species to an existing solution file.

RSNFILE_ADDSPECIES = 'casename.rsn', ! Specify name of source solution file to augment.
NEWCASE_ADDSPECIES = 'new_casename', !Specify casename of
the target (augmented) solution file.

nSpOld = , !Specify number of chemical species in source
solution.

cSpeciesNamesOld = , !Specify source species list and
order

!Process transient trace data.

PROCESS_TRACE = .F., !Post-process trace point transient
data.

TRACE_EXTRACT_OPTION = 1., !Specify how to write out
extracted trace data.

TRACEFILE = 'trace.dat', !Specify the filename of the
tracepoint file.

!Recombine time-averaged (QBar) files.

RECOMBINE_QBAR = .F., !Recombine CRUNCH solution from
partitioned files (QBar).

QBARBIN = true, !Specify CRUNCH time-averaged qbar file
format.

QBARFORM = 1., !Specify the output format of the
recombined time-averaged qbar file for plotting with
Ensight or FieldView.

!Generate solution difference report.

SOLN_DIFFERENCE = .F., !Create a solution difference
report. Valid only if both old and new restart file(s)
exist.

!Extract solution data from BC surfaces.

SURF_WRITE_DATA = .F., !Extract data from a BC surface to
a specified output file.

SURF_BCTAG = 1., !Specify surface BC tag.
SURF_NUM_VARS = 1, !Specify the number of variables to extract.
SURF_VARLIST(1) = 1, !Select variables Fluid Density
SURF_FILENAME = 'casename_surface.data', !Specify the filename of the surface data file.
SURF_APPEND = .F., !Append extracted data to the existing surface file.
/

A.13 Cavitating Global Input File Setup

The global input file requires no modification to model cavitation.

A.14 Run Cavitating Simulation

The submit job file in Section A.9 requires no modification to run a cavitating case, and is used in the exact same way it was used to run a single phase job. The cavitating simulation must be started from a single phase solution. This can be accomplished by running the cavitating case immediately after the single phase case to ensure the solution data remains in the SOLUTION directory. If this is not possible a global restart file, casename.rso, can be partitioned to perform the same function. Any restart file can be converted to a global restart file by changing the trailing information (.XXX) to .rso. To partition the global restart file must be present in the working directory, the following command is used:

PRECRUNCH.exe –i casename

If a global restart file is found, the user is given a series of options. To re-partition the solution file, the user selects the third option by inputting 3.

A.15 Post Process Cavitating Simulation

Post processing the cavitating simulation data is the same as the single phase procedure.
A.16 Repeat Cavitating Simulation Process

At this point, a converged cavitating simulation has been reached. To generate a breakdown curve, another cavitating simulation at a lower outlet pressure must be found. The following section of the cavitating module input file that defines the outlet boundary condition has been modified to define the outlet pressure to be 350 kPa, reduced from the 400 kPa of the initial cavitating simulation. The procedure of generating a single phase solution and using that as initial conditions for a cavitating simulation, and then marching down outlet pressure is the most reliable way to generate a breakdown curve.

&BCS

SUBIN_BC_TYPE(1) = 1, !Select Boundary Condition Type
SUBTYPE(1) = 1,
USUBIN(1) = 10.3493,
VSUBIN(1) = 0,
WSUBIN(1) = 0,
INFLAG(1) = 0, !Switch to outflow if flow going out of domain. Back pressure given by Static Pressure imposed.
PHIGSUBIN(1) = 0, !Cannot be zero if bubbly model is used.
TSUBIN(1) = 21.1, !Inflow static Temperature.
XKSUBIN(1) = 0, !Specify k in m^2/s^2. Code computes if not specified.
XESUBIN(1) = 0, !Specify epsilon in m^2/s^3. Code computes if not specified.
SUBIN_PROFILE(1) = .F., !Subsonic inflow read from file
BCNAME(1) = 'inlet', !Specify a name for the boundary condition.
SUBOUT_BC_TYPE(1) = 2, !Select Boundary Condition Type
A.17 Simulation Errors

Starting a cavitating simulation from scratch or decreasing outlet pressure too much often results in a numerical error, causing the simulation to stop. This section describes the errors encountered during the simulations generated for this work and potential solutions.

Should the simulation stop prematurely, the first thing to check is the simulation output file (casename.out). This is the global output file that contains results and consistency checks. If an error occurs, this file will say an error occurred and list which processor it occurred in. The next step is to find the corresponding processor output file in the OUTPUT directory named casename.outXXX, where XXX corresponds to the processor in which the error occurred. The processor output file will give more detail about what caused the solution to stop. The two most common errors encountered during the CRUNCH simulations completed for this work were temperature outside database range and quad face boundary condition.

The temperature outside database range error occurs when a mesh cell experiences a temperature lower than 14 K or higher than 30 K as the saturation property database used to calculate fluid properties only contains data between these temperatures. One method to avoid this error is to restart the simulation from a previously stable solution and continue to run at a lower CFL number for longer. The extraneous temperature could be transient and avoided with the slower convergence caused by a low CFL.

The quad face boundary condition error is generally immune to changes in CFL number. However, changing the outlet pressure has proven to help work around the error. This is a more case-dependent error, and seems to be a function of the flow coefficient and targeted NPSH defined by outlet pressure. When this error occurs, increasing or decreasing the outlet pressure has worked to alleviate the error.

Another error typical of the simulations completed for this work centers around trying to view the combined solution file from POSTCRUNCH. When trying to open the combined solution file, Paraview outputs an error reading uncompressed binary data header. This error is caused by data array offset mismatches. To remedy the error, a text editor capable of handling
large files must be used to edit the offsets. Once the file is open, look for negative offsets in the first few lines of the file before the encrypted data. Section A.18 contains the first 38 lines of an uncorrected combined solution file from POSTCRUNCH. In the unabridged file, encrypted data follows. Note the negative offset values of the vort-x through Rel_Velocity rows. To calculate the correct offsets use the difference between the single component Float64 values. For example, for all single component Float64 values in Section A.18 have a difference of 101295676 (101295672 + 4). Therefore, to fix the negative offsets, start from the last positive offset and use the difference to calculate the correct offsets. For example, the correct offset of vort-x is 2208251136 (2106955460 + 101295676). For a three component Float64 value, such as Velocity, the difference is 303887020 (303887016 + 4). Note that 303887016/3 = 101295672. So, using the correct differences to calculate the correct offsets of the preceding values, the correct offset for Velocity becomes 2613433840 (2512138164 + 101295676) and the correct offset for Cyl_Velocity becomes 2917320860 (2613433840 + 303887020).

With the information presented in this appendix, it is the hope of the author that any future users of CRUNCH working to simulate inducer performance will be at an advantage and able to avoid the issues and difficulties that slowed down the development and completion of simulations for this work.

A.18 POSTCRUNCH Output Error

```xml
<VTKFile type="UnstructuredGrid" version="0.1" byte_order="BigEndian">
  <UnstructuredGrid>
    <Piece NumberOfPoints="9855039" NumberOfCells="9523296">
      <Points>
        <DataArray type="Float64" Name="coordinates" NumberOfComponents="3" format="appended" offset="0" />
      </Points>
      <Cells>
```
<DataArray type="Int32" Name="connectivity" format="appended" offset="236520940" />
<DataArray type="Int32" Name="offsets" format="appended" offset="541266416" />
<DataArray type="Int32" Name="types" format="appended" offset="579359604" />
</Cells>
<PointData>
<DataArray type="Float64" Name="rho" format="appended" offset="790111672" />
<DataArray type="Float64" Name="P" format="appended" offset="891407348" />
<DataArray type="Float64" Name="Phi" format="appended" offset="992703024" />
<DataArray type="Float64" Name="TKE" format="appended" offset="1093998700" />
<DataArray type="Float64" Name="EPS" format="appended" offset="1195294376" />
<DataArray type="Float64" Name="mut/mul" format="appended" offset="1296590052" />
<DataArray type="Float64" Name="WDist" format="appended" offset="1397885728" />
<DataArray type="Float64" Name="TimeStep" format="appended" offset="1499181404" />
<DataArray type="Float64" Name="T" format="appended" offset="1600477080" />
<DataArray type="Float64" Name="Processor" format="appended" offset="1701772756" />
<DataArray type="Float64" Name="Soundspeed" format="appended" offset="1803068432" />

188
<DataArray type="Float64" Name="Lam. Visc" format="appended" offset="1904364108" />
<DataArray type="Float64" Name="Yplusw" format="appended" offset="2005659784" />
<DataArray type="Float64" Name="Qdotw" format="appended" offset="2106955460" />
<DataArray type="Float64" Name="vort−x" format="appended" offset="−2086716160" />
<DataArray type="Float64" Name="vort−y" format="appended" offset="−1985420484" />
<DataArray type="Float64" Name="vort−z" format="appended" offset="−1884124808" />
<DataArray type="Float64" Name="helicity" format="appended" offset="−1782829132" />
<DataArray type="Float64" Name="Velocity" NumberOfComponents="3" format="appended" offset="−1681533456" />
<DataArray type="Float64" Name="Cyl_Velocity" NumberOfComponents="3" format="appended" offset="−1377646436" />
<DataArray type="Float64" Name="Rel_Velocity" NumberOfComponents="3" format="appended" offset="−1073759416" />

</PointData>
</CellData>  </CellData>

</Piece>
</UnstructuredGrid>
<AppendedData encoding="raw">