AN ANALYTICAL AND NUMERICAL STUDY OF CAVITATION
SCALE EFFECTS IN HIGH-REYNOLDS NUMBER CIRCULAR
JET FLOWS

A Thesis in
Mechanical Engineering
by
Brian A. Edge

© 2007 Brian A. Edge

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Doctor of Philosophy

May 2007
The thesis of Brian A. Edge was reviewed and approved

* by the following:

Eric G. Paterson  
Associate Professor of Mechanical Engineering  
Thesis Advisor, Chair of Committee

Ali Borhan  
Professor of Chemical Engineering

John H. Mahaffy  
Associate Professor of Nuclear Engineering

Laura L. Pauley  
Professor of Mechanical Engineering

Mario F. Trujillo  
Research Associate, Applied Research Laboratory  
Special Member

Karen A. Thole  
Head of Mechanical and Nuclear Engineering

*Signatures are on file in the Graduate School.
Abstract

Since its earliest discovery, cavitation has proved to be a challenging topic for engineers. Engineers often seek to predict the performance of a prototype by building and testing a scale model. Model theory has traditionally worked well for predicting hydrodynamic loads, but predictions of cavitation inception from scale model tests have proved to be quite unreliable. The current theory of cavitation scaling assumes that the dimensionless cavitation number will remain constant between the model and the prototype. Experiments, however, often show that incipient cavitation numbers change significantly with length scale and the properties of the water supply. The problem is further complicated by experimental data which shows that the prototype cavitation number will sometimes increase from the model values, but at other times will decrease. Changes in the cavitation number between the model and the prototype are known as cavitation scale effects. Cavitation inception in circular jets has been the subject of recent experimental studies. These studies have found that jet flows experience significant cavitation scale effects. The data indicates that cavitation number unexpectedly increases with jet diameter.

This thesis explores the scale effects associated with cavitation inception and looks to explain the scale effects observed in jet flows. The goal of the thesis is to develop tools which can be used to predict prototype cavitation inception. As a first step, a dimensional analysis of the cavitating jet was completed. This dimensional analysis showed that the cavitation number is a function of at least eight dimensionless parameters. An analysis of these parameters shows that it is not possible to keep all of these parameters constant between the model and prototype. This means that the relative importance of the governing parameters will change with the length scale of the flow.

In order to investigate the scale effects further, a numerical code is developed to simulate the response of cavitation nuclei to a fidelity detached-eddy simula-
tion of a circular jet. The radial growth of cavitation nuclei is governed by the 
Rayleigh-Plesset equation and the dispersion of the bubbles is governed by a semi-
empirical equation of motion. This is the first time that an unsteady computational 
fluid dynamics simulation has been used in combination with the Rayleigh-Plesset 
equation to simulate cavitation inception. Results of the numerical simulations 
are consistent with previous experimental data. The scale effects observed in the 
historical data are confirmed to exist. The results also indicate that the initial nu-
clei size is the critical parameter for determining whether the incipient cavitation 
number will increase or decrease with changes in length scale.

Cavitation scale effects are also investigated from a theoretical perspective. The 
nonlinear response of nuclei bubbles to an oscillating pressure is investigated. This 
investigation leads to the development of a scaling law derived from the equilib-
rium Rayleigh-Plesset equation. The scaling law provides a method to predict the 
incipient cavitation number for a prototype flow by using the data obtained from a 
scale model. Unlike prior empirical relations and the current cavitation inception 
theory, the equilibrium scaling relation can predict and explain opposing trends 
in historical data. The equilibrium scaling law is validated by comparing with the 
numerical simulations of the jet flow and by comparing with historical hydrofoil 
data. These validations show that the equilibrium theory can predict prototype 
cavitation inception from scale model test data.
# Table of Contents

<table>
<thead>
<tr>
<th>List of Figures</th>
<th>viii</th>
</tr>
</thead>
<tbody>
<tr>
<td>List of Tables</td>
<td>xiv</td>
</tr>
<tr>
<td>Acknowledgments</td>
<td>xvii</td>
</tr>
<tr>
<td><strong>Chapter 1</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Introduction</strong></td>
<td>1</td>
</tr>
<tr>
<td>1.1 Cavitation Inception</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Cavitation Scale Effects</td>
<td>4</td>
</tr>
<tr>
<td>1.3 Cavitation Scale Effects in Circular Jets</td>
<td>9</td>
</tr>
<tr>
<td>1.4 Agenda</td>
<td>12</td>
</tr>
<tr>
<td><strong>Chapter 2</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Literature Review</strong></td>
<td>13</td>
</tr>
<tr>
<td>2.1 Nuclei Bubble Dynamics</td>
<td>13</td>
</tr>
<tr>
<td>2.2 Nuclei Bubble Transport</td>
<td>17</td>
</tr>
<tr>
<td>2.3 Numerical Studies of Rigid Bubble Motions</td>
<td>20</td>
</tr>
<tr>
<td>2.4 Numerical Studies of Cavitation Inception</td>
<td>23</td>
</tr>
<tr>
<td>2.5 Circular Jet Dynamics</td>
<td>32</td>
</tr>
<tr>
<td>2.6 Turbulence Modeling</td>
<td>39</td>
</tr>
<tr>
<td><strong>Chapter 3</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Dimensional Analysis and Cavitation Scaling</strong></td>
<td>43</td>
</tr>
<tr>
<td>3.1 Dimensional Analysis</td>
<td>43</td>
</tr>
<tr>
<td>3.2 Additional Dimensionless Groups</td>
<td>56</td>
</tr>
<tr>
<td>3.3 Categorization of Dimensionless Groups</td>
<td>57</td>
</tr>
<tr>
<td>3.4 Summary</td>
<td>59</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>6.3.1 Statistics</td>
<td>145</td>
</tr>
<tr>
<td>6.3.2 Unsteady Dynamics</td>
<td>150</td>
</tr>
<tr>
<td>6.3.2.1 Axial Velocity</td>
<td>150</td>
</tr>
<tr>
<td>6.3.2.2 Pressure</td>
<td>164</td>
</tr>
<tr>
<td>6.4 Summary</td>
<td>174</td>
</tr>
<tr>
<td><strong>Chapter 7</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Cavitation Scale Effects</strong></td>
<td>177</td>
</tr>
<tr>
<td>7.1 Equilibrium Scaling Law</td>
<td></td>
</tr>
<tr>
<td>7.1.1 Simplifications of the Rayleigh-Plesset equation</td>
<td>178</td>
</tr>
<tr>
<td>7.1.2 Origins of scale effects</td>
<td>186</td>
</tr>
<tr>
<td>7.1.3 Equilibrium scaling relation</td>
<td>188</td>
</tr>
<tr>
<td>7.1.4 Effects of non-similar liquids</td>
<td>190</td>
</tr>
<tr>
<td>7.2 Equilibrium Predictions of Jet Cavitation Inception</td>
<td>193</td>
</tr>
<tr>
<td>7.3 Numerical Test of Equilibrium Scaling Law</td>
<td></td>
</tr>
<tr>
<td>7.3.1 Operating Parameters</td>
<td>196</td>
</tr>
<tr>
<td>7.3.2 Motion of Bubbles</td>
<td>199</td>
</tr>
<tr>
<td>7.3.3 Cavitation Inception</td>
<td>202</td>
</tr>
<tr>
<td>7.4 Comparisons with Historical Data</td>
<td>207</td>
</tr>
<tr>
<td><strong>Chapter 8</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Summary, Conclusions, and Future Work</strong></td>
<td>217</td>
</tr>
<tr>
<td>8.1 Summary</td>
<td>217</td>
</tr>
<tr>
<td>8.2 Conclusions</td>
<td>218</td>
</tr>
<tr>
<td>8.3 Future Work</td>
<td>223</td>
</tr>
<tr>
<td><strong>Appendix A</strong></td>
<td></td>
</tr>
<tr>
<td>Octave FFT Script</td>
<td>228</td>
</tr>
<tr>
<td><strong>Appendix B</strong></td>
<td></td>
</tr>
<tr>
<td>Parametric Cavitation Study</td>
<td>231</td>
</tr>
<tr>
<td><strong>Bibliography</strong></td>
<td>236</td>
</tr>
</tbody>
</table>
## List of Figures

1.1 Cavitation data for 12% Joukowski hydrofoils at 0° angle of attack\[1\,2\,3\]. The theoretical $C_{P_{\text{min}}}$ for this hydrofoil is $C_{P_{\text{min}}} = -0.54$. 5

1.2 Cavitation data for NACA 16012 hydrofoils at 0° angle of attack[2]. The theoretical $C_{P_{\text{min}}}$ for this hydrofoil geometry is $C_{P_{\text{min}}} = -0.293$. 6

1.3 Cavitation number of circular jets versus the jet diameter. 10

1.4 Cavitation number of circular jets versus the jet Reynolds number. 11

2.1 Expected statistical representation of a jet (upper half) and instantaneous representation (lower half) of a high-Reynolds number circular jet. The statistics are adapted from Rouse[4] and the unsteady dynamics are adapted from Yule[5]. 33

2.2 Illustration of the grey region in a DES flow. The grey region comprises the space where the DES model transitions from its RANS mode (coarse grid) to its LES mode (fine grid). 42

3.1 Sketch of a quiescent jet and the parameters which are expected to influence the pressure at a given point within the flow. 44

3.2 Sketch of the parameters which are expected to influence the radius of a nuclei bubble. 45

3.3 Sketch of the variables which are expected to influence the total force exerted on a nuclei bubble which affect the motion of the bubble. 48

3.4 Sketch of a co-flow jet showing the additional length and velocity scales. 58

4.1 Force balance on the surface of the bubble. 73

4.2 Linear resonance frequency for nuclei bubbles from 1 to 500µm in radius. 76

5.1 Order of node numbering for a hex element. 103

5.2 Lagrangian code algorithm flowchart. 105

5.3 Example of a bubble that is within the extrema of an element, but not within the element itself. 109
5.4 Coordinate values for the nodes as defined for the element coordinate system. itself. ......................................................... 109
5.5 Definitions of variables for the inlet_domain namelist. ................ 117
5.6 Outline of all 69 meshes making up the computational domain. ... 121
5.7 Far-field c-type mesh (blue) and rectangular center far-field core mesh (red). Boundary lines for the remaining 67 meshes are visible to the left of the far-field core mesh. ...................................... 122
5.8 Meshes which make up the outer surface of the nozzle geometry. (a) exploded view of all 8 meshes with all surfaces visualized. (b) assembled view with only the nozzle outer surfaces visualized. .... 123
5.9 Meshes which make up the inner surface of the nozzle geometry. (a) assembled view of the inner surface meshes; all surfaces visualized. (b) magnified view of the nozzle exit with the front four meshes removed. ................................................................. 123
5.10 Meshes which make up the surface of the nozzle lip and the separated shear layer. (a) exploded view of all 32 meshes with all surfaces visualized. (b) assembled view with all 32 surfaces visualized. The surface of the nozzle geometry is also visualized in the background for reference. .............................................................. 124
5.11 Side view of the nozzle surface meshes and a slice through the central axis of the meshes which resolve the shear layer exiting the jet. (a) full view of the meshes which resolve the shear layer. (b) magnified view of the shear layers exiting the nozzle. ......................... 125
5.12 View of the three rectangular meshes used to resolve the flow in the center of the nozzle. (a) assembled view of the meshes and a wire frame representation of the nozzle and shear layer-resolving meshes. (b) magnified view of the central core meshes near the nozzle exit. These meshes are refined in the direction of the nozzle axis to match the level of resolution in the nozzle geometry-resolving meshes. ... 126
5.13 View of the 6 rectangular meshes used to resolve the jet wake. (a) assembled view of the 6 wake-resolving meshes along with the nozzle surface meshes in the background for reference. (b) side view of the 6 wake-resolving meshes showing the relationship to the nozzle and the shear-layer resolving meshes. ........................................... 127
5.14 View of the 10 cylindrical meshes used to resolve the flow outside of the nozzle external boundary layers and the outer regions of the jet wake. ................................................................. 128
5.15 Slice of the upper half of the cylindrical meshes which resolve the
flow outside of the nozzle external boundary layer and the outer
regions of the jet wake. The nozzle surface meshes are shown for
reference along with wire-frame representations of the shear-layer
and wake-resolving meshes. 128

5.16 Slice through the central axis of all overset domains after PEGASUS
overlap optimization. 129

5.17 Closer view of the nozzle and wake refinement meshes. 130

5.18 Closer view of the nozzle and refinement meshes. 131

5.19 Relative placement of the boundary conditions. No-slip boundary
conditions are shown in green, far-field boundary conditions are
shown in yellow, exit boundary conditions are shown in blue, and
inlet boundary conditions are shown in red. 132

5.20 Time required to read the fluid dynamics solution from disk and
broadcast to available processors with MPI versus the number of
available processors. 133

5.21 Time required to compute the bubble dynamics versus the number
of bubbles being computed per processor. 134

5.22 Terminal velocity calculations for bubbles rising due to buoyancy.
The computed data is from the bubble dynamics code and the “the-
oretical” data refers to the solution given by equation 5.21. 136

5.23 Comparison of solutions of a single bubble driven by a time varying
pressure field. The red line represents the solution obtained from the
bubble dynamics code while the black line represents the solution
obtained from Mathematica. 137

6.1 Vorticity on a slice through the central axis of the jet for the (a)
URANS simulation, (b) the DES, and (c) the ILES. The abscissa
represents $x/D_J$ coordinates while the ordinate represents $r/D_J$ co-
ordinates. 140

6.2 Axial velocity at $x/D_J = 4$ for the DES. The solid line represents
the velocity on the jet centerline and the dashed line represents the
velocity at $r/D_J = 0.5$. 144

6.3 Mean value of the axial velocity at $r/D_J = 0.5$ and $x/D_J = 4$ versus
sample size. 144

6.4 Comparison of mean velocity profiles obtained from the DES, ILES,
and RANS results at various $x/D_J$ locations. Data points are from
a 1 inch air jet at $Re = 100,000$. 147
6.5 Velocity contours on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents \( x/D_J \) coordinates while the ordinate represents \( r/D_J \) coordinates.

6.6 Comparison of computed standard deviation \( (S_U) \) of the DES and ILES results. Experimental data points are from a 1 inch air jet at \( Re = 100,000 \).

6.7 Locations of unsteady velocity and pressure measurements shown in relation to the DES vorticity field.

6.8 Velocity versus time for the DES and ILES results at \( r/D_J = 0.5 \).

6.9 Velocity versus time for the DES and ILES results along the jet axis \( (r/D_J = 0) \).

6.10 Turbulent kinetic energy (TKE) contours on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents \( x/D_J \) coordinates while the ordinate represents \( r/D_J \) coordinates. There is no TKE in the ILES solution because no subgrid turbulence model is used.

6.11 FFT of the velocity signal for the DES and ILES results at \( r/D_J = 0.5 \).

6.12 FFT of the velocity signal for the DES and ILES results along the jet axis \( (r/D_J = 0) \).

6.13 Velocity profiles at \( x/D_J = 0.01 \) for the URANS, DES, and ILES results.

6.14 Coefficient of pressure \( (C_P) \) versus time for the DES and ILES results at \( r/D_J = 0.5 \).

6.15 Dimensionless pressure \( (1/2 \text{ coefficient of pressure}) \) on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents \( x/D_J \) coordinates while the ordinate represents \( r/D_J \) coordinates.

6.16 FFT of \( C_P \) for the DES and ILES results at \( r/D_J = 0.5 \).

6.17 Coefficient of pressure data from 100 probes evenly spaced between \( 0 \leq x/D_J \leq 10 \) in the DES results.

6.18 Coefficient of pressure data from 100 probes evenly spaced between \( 0 \leq x/D_J \leq 10 \) in the ILES results.

6.19 Coefficient of pressure data from 100 probes evenly spaced between \( 1 \leq x/D_J \leq 10 \) in the ILES results.

6.20 Trajectories and pairing of coherent vortex rings which form within the DES.
### 7.1 Comparisons of numerical solutions to the full Rayleigh-Plesset equation and the equilibrium form of the Rayleigh-Plesset equation.

The black line represents the equilibrium solution and the red line represents the full R-P solution.

<table>
<thead>
<tr>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.1</td>
</tr>
<tr>
<td>(continued)</td>
</tr>
<tr>
<td>7.2</td>
</tr>
<tr>
<td>(continued)</td>
</tr>
</tbody>
</table>

### 7.2 Error between the full R-P equation and the equilibrium equation driven by a time-varying pressure.

- The mean value and frequency of the oscillating pressure are both altered to determine when the dynamic terms of the R-P equation become important.
- The fluctuating pressure has a magnitude of \( \pm 1118 \text{Pa} \) which is equivalent to 10% of \( C_{P_{\text{min}}} \) for the DES jet solution of chapter 6.

### 7.3 Percentage of 100\(\mu\text{m}\) nuclei bubbles which cavitate in the jet flow versus the cavitation number for the jet flow.

- All cavitation numbers larger than 1.38 resulted in zero cavitation events.

### 7.4 Results of 10\(\mu\text{m}\) bubbles for (a) the 0.1016\(m\) nozzle flow at \(\sigma = 1.2\), and (b) the 1.0\(m\) nozzle flow at \(\sigma = -10.28\).

### 7.5 Results of 100\(\mu\text{m}\) bubbles for (a) the 0.1016\(m\) nozzle flow at \(\sigma = 1.4\), (b) the 0.5\(m\) nozzle flow at \(\sigma = 2.1\), and (c) the 1.0\(m\) nozzle flow at \(\sigma = 4.31\).

### 7.6 Results of 100\(\mu\text{m}\) bubbles shown at relative size.

- The physical times from which the first bubbles left the nozzle are \(t = 17.1\text{s}\) for \(D_J = 1\text{m}\), \(t = 4.3\text{s}\) for \(D_J = 0.5\text{m}\), and \(t = 0.18\text{s}\) for \(D_J = 0.1016\text{m}\).

### 7.7 Cavitation events plotted versus axial location and dimensionless time for the 0.1016\(m\) jet.

- Cavitation inception is coincident with the occurrence of \(C_{P_{\text{min}}} \) in the numerical simulation. While inception is found to occur at \(x/D_J = 4.5\), a small reduction in \(\sigma\) below the value of \(\sigma_i\) initiates cavitation further upstream at \(x/D_J = 2.5\).

### 7.8 Cavitation events plotted versus axial location and dimensionless time for the 1.0\(m\) jet.

- Cavitation inception is occurs simultaneously at \(x/D_J = 2.5\) and \(x/D_J = 4.5\). The downstream location \((x/D_J = 4.5)\) coincides with the occurrence of \(C_{P_{\text{min}}} \).

### 7.9 Incipient cavitation numbers versus jet diameter.

- The solid line represents the equilibrium scale effects predicted by equation 7.16. The reference state is taken to be \(\sigma_i = 1.38\) at \(D_J = 0.1016\). This corresponds to the numerically determined incipient cavitation number from the 100\(\mu\text{m}\) parametric study.
7.10 Measured and predicted values of $\sigma_i$ for NACA 16012 hydrofoil at 2.5 inch and 5 inch chord lengths\cite{2}. The tests were conducted at $0^\circ$ angle of attack.

7.11 Cavitation data for 12% Joukowski hydrofoils at $0^\circ$ angle of attack\cite{3,1}. The theoretical $C_{P_{\text{min}}}$ for this hydrofoil is $C_{P_{\text{min}}} = -0.54$.

7.12 Equilibrium scale effects for changes in Reynolds number and length scale for the 12% Joukowski hydrofoil at $0^\circ$ angle of attack. The predictions of scale effects due to Reynolds number are given by equation 7.15 and shown by the solid red and blue lines. Predictions of scale effects due to size are given by equation 7.16 and are shown by the dashed red and blue lines. The reference states for equation 7.16 are the solid lines of the corresponding color.
## List of Tables

<table>
<thead>
<tr>
<th>Section</th>
<th>Table Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Variables governing cavitation inception along with the corresponding dimensional units and basic dimensions</td>
<td>49</td>
</tr>
<tr>
<td>3.2</td>
<td>Summary of dimensionless variables governing cavitation inception</td>
<td>53</td>
</tr>
<tr>
<td>3.3</td>
<td>Expected change in the independent II groups as the fluid is held constant but the length scale is changed.</td>
<td>55</td>
</tr>
<tr>
<td>3.4</td>
<td>Table of dimensionless groups and the category to which they are most closely associated.</td>
<td>59</td>
</tr>
<tr>
<td>4.1</td>
<td>Blended $k - \epsilon/k - \omega$ model constants.</td>
<td>66</td>
</tr>
<tr>
<td>5.1</td>
<td>Table of initial conditions for the dependent variables in the governing equations.</td>
<td>100</td>
</tr>
<tr>
<td>5.2</td>
<td>Inlet boundary conditions.</td>
<td>100</td>
</tr>
<tr>
<td>5.3</td>
<td>Exit boundary conditions.</td>
<td>101</td>
</tr>
<tr>
<td>5.4</td>
<td>Far-field boundary conditions.</td>
<td>101</td>
</tr>
<tr>
<td>5.5</td>
<td>No-slip boundary conditions.</td>
<td>101</td>
</tr>
<tr>
<td>5.6</td>
<td>Input parameters available in the <code>control</code> namelist in <code>lagrangian.nml</code>.</td>
<td>115</td>
</tr>
<tr>
<td></td>
<td>These parameters select the numerical methods and operating parameters of the numerical simulation.</td>
<td></td>
</tr>
<tr>
<td>5.7</td>
<td>Input files that are required by the code to load the fluid dynamics solutions.</td>
<td>116</td>
</tr>
<tr>
<td>5.8</td>
<td>Parameters which describe the fluid flow. These properties are set in the <code>flow_properties</code> namelist in <code>lagrangian.nml</code>.</td>
<td>116</td>
</tr>
<tr>
<td>5.9</td>
<td>Parameters used to solve the Rayleigh-Plesset equation. These properties are set in the <code>bubble_properties</code> namelist in <code>lagrangian.nml</code>.</td>
<td>117</td>
</tr>
<tr>
<td>5.10</td>
<td>Parameters which can be set to define an inlet box in which the nuclei bubbles are initially randomly distributed.</td>
<td>138</td>
</tr>
<tr>
<td>6.1</td>
<td>Table of jet velocities which correspond to jet diameter at a Reynolds number of 500,000 and assuming a standard value of kinematic viscosity for seawater.</td>
<td>141</td>
</tr>
</tbody>
</table>
7.1 Equilibrium predictions for the incipient cavitation numbers which result from various cavitation nuclei. The reference jet flow is assumed to have the following properties: \( D_J = 0.1016m \), \( U_J = 5.758m/s \), and \( C_{P_{\text{min}}} = -1.349 \). The initial bubble pressure, \( p_0 \), is found from equation 4.47, the critical pressure is calculated from equation 7.6 and \( \sigma_i \) is calculated from equation 7.11.

7.2 Equilibrium predictions for the incipient cavitation numbers which result from various cavitation nuclei. The reference jet flow is assumed to have the following properties: \( D_J = 1.0m \), \( U_J = 0.585m/s \), and \( C_{P_{\text{min}}} = -1.349 \). The initial bubble pressure, \( p_0 \), is found from equation 4.47, the critical pressure is calculated from equation 7.6 and \( \sigma_i \) is calculated from equation 7.11.

7.3 Equilibrium predictions of the incipient cavitation numbers using the smaller value of \( C_{P_{\text{min}}} \) measured from the flow at hypothetical probes placed at \( r/D_J = 0.5 \). The value of \( C_{P_{\text{min}}} \) measured from these probes is \( C_{P_{\text{min}}} = -0.82 \). The values of \( \sigma_i \) are calculated from equation 7.11.

7.4 Input parameters for the parametric study of cavitation inception.

7.5 Number of cavitation events versus the number of bubbles released into the inlet domain of the jet flow. All simulations were for a 0.1016m nozzle at \( Re = 500,000 \), \( C_{P_{\text{min}}} = -1.349 \), and \( \sigma = 1.35 \).

7.6 Number of cavitation events versus the number of sub-time steps used in the R-P and EoM dynamics. All simulations were for 10,000 bubbles of 100µm radius released in a flow with the following properties: \( D_J = 0.1016m \), \( Re = 500,000 \), \( C_{P_{\text{min}}} = -1.349 \), and \( \sigma = 1.35 \).

7.7 Values of the inertia parameter, \( A \), for various bubble sizes and jet sizes (\( Re_J = 500,000 \)).

7.8 Values of the scaled particle settling velocity, \( W_s \), for various bubble and jet sizes (\( Re_J = 500,000 \)).

7.9 Numerically obtained cavitation inception numbers compared with equilibrium predictions from tables 7.1 and 7.2.

7.10 Comparison of the numerical solutions and the predictions given by the equilibrium scale effects from equation 7.16. The reference states for the equilibrium scale effects are the numerical results for \( D_J = 0.1016 \) listed in the second column.

B.1 Cavitation results for 10,000 nuclei bubbles with an initial radius of \( a_0 = 100\mu m \). The diameter of the jet is \( D_J = 0.1016m \), the Reynolds number is \( Re = 500,000 \), and the minimum coefficient of pressure is \( C_{P_{\text{min}}} = -1.349 \).
B.2 Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 100 \, \mu m$. The diameter of the jet is $D_J = 0.5 \, m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$. 

B.3 Cavitation results for 10,000 nuclei bubbles with an initial radius of $a_0 = 100 \, \mu m$. The diameter of the jet is $D_J = 1.0 \, m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$.

B.4 Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 50 \, \mu m$. The diameter of the jet is $D_J = 0.1016 \, m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$.

B.5 Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 50 \, \mu m$. The diameter of the jet is $D_J = 1.0 \, m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$.

B.6 Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 10 \, \mu m$. The diameter of the jet is $D_J = 0.1016 \, m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$.

B.7 Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 10 \, \mu m$. The diameter of the jet is $D_J = 1.0 \, m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$. The term “All Deactivated” indicated that the bubble radius became unstable and $a \rightarrow \infty$. When this occurs, the EoM becomes unstable and the bubbles advect out of the domain (i.e. $\vec{x} \rightarrow \infty$). The code reports that the bubble has been deactivated.
Acknowledgments

This research was conducted at the Applied Research Laboratory at The Pennsylvania State University. Financial support for this thesis was provided by Diane Segelhorst at the Naval Sea Systems Command (NAVSEA). Computational resources for this work were provided by the Department of Defense High Performance Computing Modernization Program. Without the generous support of these sponsors, the work presented in this thesis would not have been possible.

I would like to acknowledge my adviser, Dr. Eric Paterson, for his patience, understanding, and guidance. During my time at Penn State he has treated me as though I were a colleague. His instruction and guidance were always positive and encouraging. For this, I am very grateful. I would like to give special thanks to Dr. Mario Trujillo. He has given a significant amount of his personal time to educate and mentor me through this work. His persistent and probing questions have taught me how to think critically and question my own research. I owe much of my academic development during the last few years to Dr. Trujillo. I would also like to acknowledge the staff and students at the Garfield Thomas Water Tunnel. Your friendship has meant a great deal to me over the last few years. I wish you all the best in the future.

I would like to acknowledge my wife, Lisa. Without her encouragement and support, I would not have been able to complete this work. A Ph.D. thesis requires a significant sacrifice, not only of yourself, but also of those who are close to you. Thank you for enduring these sacrifices with patience and understanding.

Finally, I would like to thank my parents. They provided a childhood where I discovered the joy of learning, the necessity of hard work, and the patience required to finish a project of this magnitude. Without these attributes, I would not have been prepared for the challenges of writing this thesis.
Chapter 1

Introduction

1.1 Cavitation Inception

Cavitation is the occurrence of vapor and gas-filled voids which form in response to the dynamic actions of a flowing liquid. The first detectable voids which form in a liquid are referred to as incipient cavitation. These incipient cavities are detected when they reach a visually observable size or when they radiate sufficient sound energy to be detected above background noise. Engineers typically try to avoid the occurrence of cavitation since it can lead to reduced efficiency, surface erosion, and noise.

The key to avoiding cavitation is to predict when and where it will occur in a liquid flow. Once the conditions which produce cavitation are known, the design or operating parameters can be changed to avoid this state. Traditional cavitation theory expects that inception will occur when the minimum pressure in the flow reaches the vapor pressure of the liquid. This means that engineers must know the minimum pressure in a liquid flow to predict cavitation inception. Unfortunately, the minimum pressure can be difficult to measure. Instead of trying to measure the minimum pressure, cavitation inception is correlated to an easily measured reference pressure. Cavitation experiments are typically conducted at constant operating velocity and consist of reducing the background pressure until incipient cavitation is observed (either visually or acoustically). The reference pressure is recorded and used to define an incipient cavitation number, $\sigma_i$. The incipient cavitation number is a dimensionless parameter which represents the ratio of the
difference between the background pressure and the vapor pressure at inception to the maximum pressure differential which can result from the reference velocity. The incipient cavitation number is defined as,

$$\sigma_i = \frac{p_{\infty,i} - p_v}{\frac{1}{2} \rho U_{\infty}^2},$$

(1.1)

where $p_{\infty,i}$ is the reference value of the pressure when cavitation occurs, $p_v$ is the vapor pressure of the liquid, $\rho$ is the density of the liquid, and $U_{\infty}$ is the characteristic velocity of the flow.

If cavitation occurs at the instant when the minimum pressure, $p_{\text{min}}$, reaches the vapor pressure, then $\sigma_i$ will be,

$$\sigma_i = -C_{P_{\text{min}}},$$

(1.2)

where $C_{P_{\text{min}}}$ is the minimum coefficient of pressure defined as,

$$C_{P_{\text{min}}} = \frac{p_{\text{min}} - p_{\infty}}{\frac{1}{2} \rho U_{\infty}^2}.$$

(1.3)

For incompressible flows without significant gravitation effects the value of $C_{P_{\text{min}}}$ is a function of the geometry of the flow and the Reynolds number of the flow. As long as the flow remains an incompressible liquid (e.g. no phase change) the value of $C_{P_{\text{min}}}$ is independent of the magnitude of $p_{\infty}$ and $p_{\text{min}}$. The Reynolds number of the flow is defined as,

$$Re = \frac{U_{\infty} L_{\infty}}{\nu},$$

(1.4)

where $L_{\infty}$ is the characteristic length scale of the flow and $\nu$ is the kinematic viscosity of the liquid. Physically, the Reynolds number represents the ratio of the inertia forces to the viscous forces in the liquid flow. Since $C_{P_{\text{min}}}$ is a function of the geometry and Reynolds number, a scale model test operating at the same Reynolds number as a larger prototype satisfies,

$$C_{P_{\text{min}},m} = C_{P_{\text{min}},p},$$

(1.5)

where $C_{P_{\text{min}},m}$ and $C_{P_{\text{min}},p}$ are the minimum coefficients of pressure for the model.
and prototype, respectively. Please note that the term prototype is used to describe the larger geometry which the scale model represents. If equation 1.5 holds true then equation 1.2 gives,

\[ \sigma_{i,m} = \sigma_{i,p}, \]  

(1.6)

where \( \sigma_{i,m} \) and \( \sigma_{i,p} \) are the incipient cavitation numbers for the model and prototype flow, respectively. Equations 1.2, 1.5, and 1.6 provide the basis for using scale model tests to predict prototype cavitation performance. The conditions which satisfy equation 1.5 are known as dynamic similarity. Dynamic similarity for an incompressible liquid flow without significant gravitational effects requires that both Reynolds number and geometric similarity are maintained. Reynolds number similarity ensures that the ratio of the inertia forces to the dynamic forces is the same in the model and prototype flows. Geometric similarity requires that all aspects of the model geometry are a scaled representation of the prototype geometry. This includes surface roughness which is difficult to achieve in practice.

Reynolds number similarity is also difficult to achieve for high Reynolds number prototype flows. Reynolds number similarity requires that the Reynolds number of the model flow must be equal to the Reynolds number of the prototype flow. Using the definition of the Reynolds number in equation 1.4 Reynolds number similarity will be achieved if,

\[ \frac{U_{\infty,m} L_{\infty,m}}{\nu_m} = \frac{U_{\infty,p} L_{\infty,p}}{\nu_p}, \]  

(1.7)

where the subscripts “m” and “p” signify that the variables are associated with either the model or prototype, respectively. Since it is common to test both the model and prototype in water, the value \( \nu \) is not expected to change significantly. Letting \( \nu \) be a constant, the model velocity must be equal to,

\[ U_{\infty,m} = U_{\infty,p} \left( \frac{L_{\infty,p}}{L_{\infty,m}} \right). \]  

(1.8)

Equation 1.8 shows that the model velocity must be larger than the prototype velocity by the ratio \( L_{\infty,p}/L_{\infty,m} \) (assuming that the prototype is larger than the model). For large Reynolds number problems where \( U_{\infty,p} \) and/or \( L_{\infty,p} \) are very large, Reynolds number similarity will require that the model velocity be very
large. It is common that the required model velocity cannot be obtained with the test apparatus. Because of this, many scale model tests are performed at lower Reynolds numbers than the prototype flow. Since $C_{P_{\text{min}}}$ is a function of the Reynolds numbers, $C_{P_{\text{min}},m}$ may not equal $C_{P_{\text{min}},p}$. However, if $C_{P_{\text{min}}}$ is a weak function of the Reynolds numbers spanned by the model and prototype flow, then $C_{P_{\text{min}},m} \sim C_{P_{\text{min}},p}$ and $\sigma_{i,m} \sim \sigma_{i,p}$.

### 1.2 Cavitation Scale Effects

The cavitation theory presented so far indicates that the prediction of cavitation inception for a prototype should be straightforward. As long as $C_{P_{\text{min}},p}$ is known, or as long as a dynamically similar model test can be conducted, engineers should be able to predict the value of $\sigma_{i,p}$. Even when a dynamically similar model cannot be achieved, as long as $C_{P_{\text{min}},m} \sim C_{P_{\text{min}},p}$ the value of $\sigma_{i,p}$ should be approximately known. Unfortunately, in practice it has not been possible to predict $\sigma_{i}$ from the value of $C_{P_{\text{min}}}$ or to use the value of $\sigma_{i,m}$ to predict $\sigma_{i,p}$. Experimental data often shows that $\sigma_{i} \neq -C_{P_{\text{min}}}$ and that $\sigma_{i}$ is not constant for dynamically similar flows\cite{2,3,7,8}. Even tests of the same geometry conducted in different test facilities give significantly different results for $\sigma_{i}$ when tested at the same Reynolds numbers\cite{9,10}.

The discrepancies with equation 1.6 which are typically observed in experiments are known as cavitation scale effects. The occurrence of scale effects has motivated a considerable amount of research in the past 50 years. Figures 1.1 and 1.2 show measured values of $\sigma_{i}$ for hydrofoils of different size. Even when the hydrofoils are tested at the same Reynolds number, the cavitation numbers can be significantly different. This seems to indicate that the scale of the flow is also important. However, no correlations between $\sigma_{i}$ and other dimensionless groups have been found which fully explain these trends with scale. The problem is that conflicting trends are often observed. For example, the Joukowski hydrofoil shown in figure 1.1 shows that $\sigma_{i}$ decreases as the size of the hydrofoil increases. The NACA 16012 hydrofoil shown in figure 1.2 shows the opposite trend where $\sigma_{i}$ increases with the size of the hydrofoil. The trends with Reynolds number are also conflicting. The Joukowski hydrofoil shows an increase in $\sigma_{i}$ with Reynolds number while the
NACA 16012 hydrofoil shows a decrease in $\sigma_i$ with Reynolds number. While the shape of the hydrofoils are different, it is curious that they would result in vastly different trends with scale and Reynolds number. The opposing trends such as these have made it difficult for hydrodynamic engineers to predict $\sigma_{i,p}$ from $\sigma_{i,m}$.

![Cavitation data for 12% Joukowski hydrofoils at 0° angle of attack][1]. The theoretical $C_{P_{\min}}$ for this hydrofoil is $C_{P_{\min}} = -0.54$.

While the opposing trends have proved to be difficult to explain and predict, it is not entirely surprising that $\sigma_i \neq -C_{P_{\min}}$. The cavitation theory presented so far neglects many sources of error and uncertainty which are associated with the causes and detection of cavitation inception. Possible sources of error and uncertainty include,

- method of detection (visual or acoustic)
- definition of cavitation
- “liquid quality” – or cavitation occurring when $p_{\min} \neq p_v$
- Reynolds number effects

[1]: cavitation.png
Figure 1.2: Cavitation data for NACA 16012 hydrofoils at 0° angle of attack[2]. The theoretical $C_{P_{\text{min}}}$ for this hydrofoil geometry is $C_{P_{\text{min}}} = -0.293$.

- geometry effects (e.g. surface roughness)
- turbulence
- and residence time.

The first two subjects on the list are related to the detection of cavitation inception. It has been shown that the method of detection can change the measured cavitation inception number[11, 12]. Methods of detection include visual detection and acoustic detection. Visual detection means that voids which contain vapor or gas must expand to a size where they can be seen. Acoustic detection means that voids in the fluid have begun to change size rapidly and are emitting sound waves which can be detected. Experiments have shown that sound emissions associated with cavitation inception are often detected at higher background pressures than those associated with visually-detected cavitation inception. Cavitation which is detected acoustically prior to visual observation is known as “sub-visual cavitation”.
While the method of detection is known to affect $\sigma_i$, the definition of cavitation inception can also introduce errors. Regardless if cavitation is detected visually or acoustically it is reasonable to expect that different experimental setups with different sensitivity in detecting cavitation would result in different values of $\sigma_i$. When using a strobe light and the unaided eye, cavities in a flowing liquid are reported to be visible when they grow to a diameter of at least 1mm\cite{13}. Other cavitation experiments which use holographic methods have been shown to detect nuclei which are less than 100$\mu$m in diameter with a measurement precision of $\pm 2\mu m$\cite{14}. Another problem with inception is that it is often transient and characterized by cavities which grow and collapse very rapidly\cite{15}. This transient nature makes it difficult to detect the rate at which it occurs and makes the definition subjective. Testing has shown that uncertainty in $\sigma_i$ from one test to the next, or from one operator to another, is on the order of $\% 15$\cite{16}. While these first two sources of uncertainty will result in scatter in the measured values of $\sigma_i$ from test-to-test and from one testing facility to the next, it does not account for the scale effects shown in figures 1.1 and 1.2. Since each data point was recorded from the same test setup, a uniform method of detecting cavitation and a uniform definition of cavitation were used to obtain each of the data points.

Another uncertainty which may explain differences in $\sigma_i$ from different test facilities is “liquid quality”. The liquid quality is a generic term which is used to describe the fact that different water supplies will contain differing numbers and sizes of nucleation sites. Liquid quality is also used to describe the differing levels of dissolved gas which a liquid may contain. The implication for liquid quality is that the critical pressure at which inception occurs can be altered. A liquid which has been treated to remove the larger nucleation sources has been found to withstand considerable tensions prior to phase change or the formation of a cavity in the liquid. Tension within a liquid is defined as the difference between the vapor pressure and the local pressure, $p$, when the local pressure is less than the vapor pressure. To observe cavitation in such a liquid, the minimum pressure must be brought to $p_{\text{min}} < p_v$. This is an unstable state and requires special treatment of the liquid. When cavitation does occur in a liquid under tension, cavities tend to grow explosively without bound by the rapid conversion of liquid to vapor. This form of cavitation is often called \textit{vaporous cavitation}\cite{3}. 
When significant quantities of nuclei bubbles and particles with pockets of trapped gas exist in the liquid, cavitation is observed to occur at $p_{min} > p_v$. In these cases, the pockets of gas will expand in response to changing pressure in the liquid. As the background pressure is reduced, the gas will expand until it is observed visually or until it becomes unstable and emits sound. This type of cavitation is called *pseudo cavitation* or *gaseous cavitation*. It is well understood that the liquid quality is an important factor in cavitation inception\[17\]. However, defining what liquid quality means is still a matter of debate. For example, some articles cite the number and size of nuclei bubbles when discussing water quality\[10, 11\] while others discuss the dissolved gas content\[18, 19, 12\]. Regardless of the definition, measuring water quality is a difficult task and is not always reported in cavitation studies. While liquid quality can explain the scatter in the value of $\sigma_i$ from test-to-test and from one testing facility to the next, it does not account for the scale effects shown in figures 1.1 and 1.2. These tests were carefully conducted in testing facilities where the liquid quality could be brought to the same state between tests.

While quantifying Reynolds number effects has already been addressed by the traditional theory of cavitation inception, there is still significant uncertainty involved in measurement. It is common for high Reynolds number problems that the model flow cannot operate at the same Reynolds number as the prototype. If $C_{P_{min}}$ is known for both flow conditions, then equation 1.2 is expected to provide at least an indication of how $\sigma_i$ will change. However, since the model cannot be operated at the prototype Reynolds number, the value of $C_{P_{min},p}$ is not likely to be known. Also, measuring $C_{P_{min}}$ in a transient flow is quite difficult which means that even $C_{P_{min},m}$ may not be known. Because $C_{P_{min}}$ and its relationship with Reynolds number is not known, the theoretical change in $\sigma_i$ is unknown. The value of $C_{P_{min}}$ for figures 1.1 and 1.2 are theoretical values and were not directly measured. The theoretical values of $C_{P_{min}}$ are usually obtained from inviscid assumption and therefore cannot account for changes in $C_{P_{min}}$ with Reynolds number which may occur in the actual flow. While discrepancies between the theoretical and actual value of $C_{P_{min}}$ may explain the change in $\sigma_i$ with Reynolds number, it still does not explain why $\sigma_i$ would change with the scale of the test. If the flows are geometrically similar, Reynolds number similarity should ensure that $C_{P_{min},m} = C_{P_{min},p}$ and therefore that $\sigma_{i,m} = \sigma_{i,p}$. However, geometric effects such
as relative changes in surface roughness have been shown to affect the value of $\sigma_i$. These geometric scale effects remain as a possibility for the scale effects observed in figures 1.1 and 1.2.

Turbulence in the flow will cause fluctuations in the pressure. These perturbations can cause the pressure in a small turbulent eddy to drop below the critical value required for cavitation inception. The small physical size and large velocity of the scale model flow make detection of the resulting cavitation very difficult. The prototype, however, is much larger in size and will have a smaller reference velocity. This means that detecting cavitation associated with the turbulent eddy may be easier. The result is that cavitation may be detected at different values of $\sigma_i$ in the model and prototype flows.

The amount of time that a nucleation source is subjected to a sufficiently low pressure may also have an effect on the value of $\sigma_i$. In the smaller model flow, nucleation sources will spend less time in the presence of a low pressure than they will in the larger prototype flow. This means that nuclei will have less time to expand to a size where they can be observed. This theory still does not explain the conflicting trends with Reynolds number since one would assume that higher velocities (i.e. higher Reynolds numbers) would give the nuclei less time to expand. If the amount of time a nuclei spent in the presence of a low pressure were important, one would expect to see an inverse relationship between $\sigma_i$ and Reynolds number. Figure 1.1 shows a case where $\sigma_i$ increases as Reynolds number increases. While temporal effects may still be important, the Joukowski hydrofoil data indicates that this effect is probably of secondary importance as $\sigma_i$ is increasing as the Reynolds number increases (an increasing value of $\sigma_i$ indicates that cavitation is occurring at larger background pressures). If temporal effects were of first order importance we would expect to see the opposite trend where cavitation number would increase as the Reynolds number (velocity) decreased. This would give the nuclei more time to expand to a visible size.

1.3 Cavitation Scale Effects in Circular Jets

For more than 50 years, cavitation inception in circular jets has been a topic of a considerable amount of research. Results of cavitation studies indicate that circular
jets also exhibit significant scale effects with the general trend that $\sigma_i$ increases with the jet diameter, $D_J$. Figure 1.3 summarizes the results of several cavitation studies of circular jets. The correlation with jet diameter is not a Reynolds number effect. The incipient cavitation number is plotted versus the jet Reynolds number in figure 1.4. This figure indicates that there is little correlation between $\sigma_i$ and Reynolds number for the jet, a fact which has been pointed out by others [20].

![Figure 1.3: Cavitation number of circular jets versus the jet diameter.](image)

Based on the theory of cavitation inception from equation 1.2, the only reason why cavitation inception should change with $D_J$ is that the Reynolds numbers of the flow is also changing. Since there are no strong correlations with Reynolds number, the reason for the relation between $\sigma_i$ and $D_J$ remains a mystery. The previously listed uncertainties may provide some clues as to what is occurring. The method of detection and definition of cavitation are clearly important. The holographic detection used by Ran and Katz and Gopalan and Katz along with the acoustic detection from Straka et al. result in larger cavitation numbers than the other studies which use visual detection. The visual studies detected cavitation
at $x/D_J > 5$ which was further downstream than the sub-visual studies which detected cavitation at $x/D_J < 5$. Regardless of the method of detection, the increase in $\sigma_i$ with $D_J$ is still apparent.

A numerical study of jet flows by Cerutti et al.[21] indicates that the initial shear layer thickness is very important to value of $C_{P_{min}}$. This observation has led to the suggestion that geometry effects may be contributing to the observed correlation between $\sigma_i$ and $D_J$. The theory is that experimental setups allow larger jets to have thinner initial shear layers, and therefore smaller values of $C_{P_{min}}$. Unfortunately this cannot be confirmed since the values of the initial shear layer thickness were not reported by most of the prior jet studies. Finally, residence time could be a factor since larger jets will give nucleation sources more time to grow. This would allow cavitation inception to be observed at larger values of $\sigma_i$ in larger jets.
1.4 Agenda

This thesis will explore the origins of cavitation scale effect and try to explain the scale effects observed in jets. The first task which will be undertaken is a dimensional analysis of the cavitation inception in jets. This will provide a list of the dimensionless parameters which govern cavitation inception. These dimensionless parameters will be investigated to gain insight into possible causes of the cavitation scale effects. Following the dimensional analysis the governing equations for the computational fluid dynamics, the radius of the cavitation nuclei, and the dispersion of the cavitation nuclei will be derived. The derivation of these equations will provide an understanding of the limitations of the equations and insight into the parameters which are important to cavitation inception.

The problem of cavitation inception in jets will also be examined through a numerical simulation. A Numerical code is developed which simulates the radial growth and dispersion of the cavitation nuclei in an unsteady computational fluid dynamics simulation of the jet flow. A fidelity detached-eddy simulation of a jet is developed and used with the bubble dynamics code to investigate cavitation inception in different size jets. The results will be compared with historical data to determine if the simulations of cavitation inception predict the same cavitation scale effects which are found in experimental data. Several different nuclei sizes will also be examined to determine the importance of the liquid quality.

The governing equations for the cavitation nuclei are examined to determine if cavitation scale effects can be predicted. Simplified solutions to the nonlinear equations for the bubble size will be sought. Scaling relations will be derived from the simplified solutions and validated by comparing with the numerical simulations of the jet and historical hydrofoil data.
Chapter 2

Literature Review

2.1 Nuclei Bubble Dynamics

Studies into the mechanism which causes incipient cavitation have revealed that it can occur from one of two different mechanisms: homogeneous nucleation and heterogeneous nucleation[22, 9]. Homogeneous nucleation is a thermodynamic process which involves thermal motions of the molecules within the fluid. These thermal motions lead to temporary microscopic voids in the fluid which can rupture the liquid and lead to incipient cavitation (or boiling) under suitable conditions[9]. Homogeneous cavitation is typically associated with cavitation in highly pure liquids and a large liquid tensions. Heterogeneous nucleation is the result of the growth of microscopic nuclei bubbles and pockets of gas which are trapped in the crevices of motes and/or the liquid container. The pressures associated with heterogeneous nucleation are much closer to the vapor pressure of the liquid than the pressures associated with homogeneous nucleation. Therefore real world incipient cavitation is considered to be a heterogeneous nucleation phenomena where the incipient cavitation voids are the result of the dynamics of small gas and vapor filled nuclei bubbles responding to a local reduction in the fluid pressure. Detailed studies of the heterogeneous cavitation nuclei have shown that microscopic nuclei bubbles are more prevalent than motes (particles in suspension in a fluid) in most engineering flows[22]. As such, most of the theoretical understanding of cavitation inception is based upon the dynamics of small spherical gas and vapor filled bubbles.

Among the first to consider the dynamics of a spherical cavity in a liquid
were Besant[23] and Rayleigh[24]. They studied the time required for an empty, spherical cavity to collapse within a liquid. Plesset[25] modified their formulation to account for a time-varying pressure field within the liquid and considered a cavity which is not empty, but filled with vapor. Noltingk and Neppiras[26, 27] then added terms to account for an adiabatic gas within the bubble and Poritsky[28] added terms to account for viscosity within the liquid. The result of these contributions is the governing equation for a spherical gas and vapor-filled bubble. This equation is known as the Rayleigh-Plesset (R-P) equation and is a second-order nonlinear ordinary differential equation (ODE) with the form,

\[ a\ddot{a} + \frac{3}{2} (\dot{a})^2 + 4\nu \frac{\dot{a}}{a} + \frac{2S}{\rho a} - \frac{p_0}{\rho} \left( \frac{a_0}{a} \right)^3 n = \frac{p_v - p}{\rho}, \quad (2.1) \]

where \( a \) is the bubble radius, \( \nu \) is the liquid kinematic viscosity, \( S \) is the surface tension for the bubble interface, and \( n \) is the polytropic exponent. The value of the polytropic constant is 1 when the bubble undergoes isothermal compression/expansion or the value of the gas constant for adiabatic compression/expansion.

Some of the implicit assumptions involved in deriving equation (2.1) are that the bubble remains spherical at all times, the compression process is internally reversible, and no mass is transferred across the bubble interface. More information regarding the assumptions required to derive equation (2.1) will be provided in section 4.2. In the mean time, it is important to note that the assumptions behind the R-P equation limit its applicability. For instance, during a violent collapse of a bubble, the compression process is highly irreversible with significant energy losses due to heat transfer, acoustic radiation, as well as electromagnetic radiation in the form of light when a sonoluminescing collapse occurs. Even though the R-P equation is not valid for all dynamics which a nuclei bubble can experience, it has been shown to do a good job of reproducing the qualitative dynamics of a bubble[29]. More sophisticated models have been developed which attempt to account for effects such as heat and mass transfer as well as liquid compressibility. These models add a considerable amount of mathematical complexity and only provide small improvements over the R-P equation in most regimes[29]. While these equations do perform much better during a violent collapse, these events do not typically occur in reality because of shape instabilities which tend to destroy
the bubble during a violent collapse. Due to their added mathematical complexity and negligible improvement in accuracy, the more sophisticated models are not typically used in studies of bubble motions.

At this time there is no known general solution for the R-P equation. Solutions are possible if small perturbations are assumed, which reduces the equation to a weakly nonlinear oscillator\[30\]. This form can be linearized which results in an equation with the form of a harmonic oscillator\[29\]. From this form, the linear resonance frequency (often called the natural frequency) of a bubble can be obtained. This frequency is \[30\],

\[
\omega = \sqrt{\frac{3n}{\rho a^2} \left( p - p_v + \frac{2 S}{a} \right) - \frac{2 S}{\rho a^3}}, \tag{2.2}
\]

where \(\omega\) is the angular resonance frequency in radians per second. It is apparent from equation\[2.2\] that the natural frequency is inversely proportional to the bubble size. This is important because typical bubble natural frequencies are often much higher than the frequencies associated with the energy-containing eddies in the turbulent flow. However, as a bubble grows in response to low pressure in the liquid, it becomes receptive to lower frequencies and eventually may become receptive to frequencies corresponding to the frequency of the turbulent fluctuations in the liquid.

Study of the linear equations has also revealed that the significant damping mechanisms for an oscillating bubble are energy loss through heat transfer and acoustic radiation. Unfortunately, neither of these mechanisms of energy loss is taken into account by the Rayleigh-Plesset equation. The only energy loss accounted for in the Rayleigh-Plesset equation is viscous dissipation. Analysis has revealed that only very small bubbles \((O(1\mu m))\) are dominated by viscous damping\[30\]. Because of this, the R-P equation only predicts the immediate response of a bubble to an unsteady pressure signal and significantly under-predicts the damping that an actual bubble will experience. The under-prediction of damping is only a problem when significant inertial effects lead to vibratory oscillations about the bubble equilibrium state. These oscillations typically occur after a collapse, or other strong nonlinear behavior.

While the linearized representation of the nuclei bubble has illuminated many
aspects of bubble dynamics, only very small oscillations about the bubble equilib-
rium radius can be approximated as linear. For all other oscillations, the radius of
the bubble must be evaluated through numerical solutions of the R-P equation[29].
The R-P equation resolves the complex nonlinear behavior of the bubble radius in
response to a time-varying pressure. The nonlinear behavior of the bubble radius
is receptive to both the magnitude and frequency of pressure oscillations within
a flow. The dynamics of the bubble radius are also sensitive to the magnitude of
the polytropic constant. In order to determine the proper value of the polytropic
constant, Ran and Katz[14] conducted a series of calibration experiments for the
polytropic constant, \( n \). Their results indicated that an assumption of isothermal
behavior (\( n = 1.0 \)) is appropriate for cavitation nuclei bubbles in water.

The nonlinear behavior of a bubble (and the R-P equation) can be classified
into two groups: stable cavity dynamics and transient cavity dynamics. Stable cavi-
ty dynamics are those in which the radial change of the nuclei bubble is inversely
proportional to the change in the background pressure. With stable cavity dy-
namics the nuclei bubble response will appear to be “stable” with bounded radial
oscillations which have a frequency equal to the frequency of the external pressure
oscillations. Transient cavity dynamics are characterized by inverse, but dispro-
portionate, changes in the bubble radius. These changes in radius often amount to
several times the initial bubble size. The radial growth is then followed by a rapid
and violent collapse where the radial velocity of the bubble interface can reach,
or even exceed, the speed of sound in the fluid. Experimentally, transient cavity
dynamics typically lead to a violent breakup of the bubble into a series of much
smaller bubbles due to the growth of shape instabilities. Transient cavities are
most easily identified by plotting \( d(a/a_0)/dt \) versus \( a/a_0 \). The transient cavities
correspond to the plot growing toward \( \pm \) infinity along the \( d(a/a_0)/dt \) axis.

It has been shown that the driving pressure (\( p_v - p_\infty(t) \)) is the crucial parameter
to determining the bubble response[29]. While much emphasis has been placed on
understanding bubble response to driving pressure frequencies which are near or
above the bubble linear resonance frequency[30, 22], little attention has been given
to the relatively low frequency pressure fluctuations which are more characteristic
of hydrodynamic turbulence. While bubble response typically falls into the stable
cavity group under relatively low frequency forcing, these observations have been
made for relatively small amplitude oscillations which are typical for acoustically driven bubbles. Furthermore, as a bubble grows in response to a low pressure event, the frequencies to which the bubble is susceptible to transient cavity growth, will decrease. For this reason, the susceptibility of cavitation nuclei to pressure fluctuations representative of hydrodynamic turbulence should be evaluated closer.

This summary of the bubble dynamics literature indicates that nonlinear behavior in the bubble radius should be expected in response to all but very small time-varying pressure fluctuations. The dynamics must be resolved through numerical solution of the governing equation as no analytical solutions exist for the full nonlinear problem. While neglecting some important physics, the R-P equation has proved to be a good model for the dynamic response of spherical bubbles to a time varying pressure field. Since the response of the nuclei bubbles is nonlinear with radius, the past history of the bubble can be important to the response to a local pressure fluctuation. Thus we must also consider the path which a bubble has traversed through the flow.

### 2.2 Nuclei Bubble Transport

The dynamics of a bubble in a hydrodynamic flow are not limited to the radial response of the bubble to changes in the local pressure. The bubble will also be advected by the local fluid with a velocity which is not identical to that of the local fluid due to inertial effects and buoyancy. The difference between the bubble velocity and the fluid velocity at the center of the bubble (assuming that the bubble were not present) is called the slip velocity. The existence of a slip velocity introduces a fundamental question regarding the behavior of the interface between the liquid and the gas within the bubble. Liquid and gas interfaces are characterized by a balance of shear stress in the two fluids[31]. When the bubble moves at a different velocity than the surrounding fluid, the balanced shear stress boundary condition results in an internal circulations within the bubble and a nonzero velocity at the liquid and gas interface. In the limit of creeping flow and negligible gas viscosity within the bubble, the steady-state drag force on a bubble is 2/3 that of a solid sphere. However, experiments of gas bubbles in tap water indicate that the drag of small spherical bubble rising in water is the same as that
of solid sphere\textsuperscript{32, 33}. These experiments indicate that the gas and liquid interface acts as a no-slip boundary. This behavior has been attributed to surfactants which collect on the bubble surface and which have the effect of immobilizing the interface between the exterior liquid and interior gas/vapor mixture. This surfactant shell has also been attributed to enabling the nuclei bubble to persist for long periods of time when theory shows that the gas within the bubble should have dissolved into the surrounding liquid\textsuperscript{22, 9}.

The equation of motion (EoM) for a rigid sphere in a non-uniform creeping flow was derived by Maxey and Riley\textsuperscript{34}. In the limit of creeping flow, the governing equations behave linearly making the total force on the sphere a linear combination of the various body forces and surface forces which act on the sphere. These forces are,

\[
m_b A_b = F_{\text{gravity}} + F_{\text{external flow}} + F_{\text{added mass}} + F_{\text{drag}} + F_{\text{history}} \tag{2.3}
\]

where \(F_{\text{gravity}}\) is the buoyancy force which exists because of the difference in density between the sphere and the surrounding fluid. The stresses in the fluid flowing around the bubble are accounted for in \(F_{\text{external flow}}\). The added mass force, \(F_{\text{added mass}}\), is the inertial force associated with accelerating the liquid which surrounds the sphere and is “pulled along” when the sphere is accelerated. The drag force, \(F_{\text{drag}}\), accounts for the viscous shear stress which arises when the sphere velocity is different from the surrounding liquid velocity while the Basset history term, \(F_{\text{history}}\), accounts for transient forces which arise when the bubble is accelerated. The full form of the EoM as derived by Maxey and Riley is,

\[
\frac{m_b}{m_f} \frac{dV_i}{dt} = (m_b - m_f) g_i + m_f \frac{Du_i}{Dt} - \frac{1}{2} \frac{m_f}{m_b} \frac{d}{dt} \left\{ V_i - u_i - \frac{1}{10} \frac{a^2 \nabla^2 u_i}{\mu} \right\} \\
- 6\pi \frac{a \mu}{\nu} \left( V_i - u_i - \frac{1}{6} \frac{a^2 \nabla^2 u_i}{\mu} \right) - 6\pi \frac{a^2 \mu}{\nu} \int_0^t d\tau \left( \frac{d/d\tau \left( V_i - u_i - \frac{1}{6} \frac{a^2 \nabla^2 u_i}{\mu} \right)}{\sqrt{\pi \nu (t - \tau)}} \right) \tag{2.4}
\]

where \(m_b\) is the mass of the sphere, \(m_f\) is the mass of the fluid displaced by the sphere, \(V_i\) is the velocity of the sphere, \(u_i\) is the mass of the fluid at the location of the sphere if the sphere did not exist. The radius of the sphere is denoted by the symbol \(a\) while the dynamic viscosity and kinematic viscosity of the fluid is denoted by the symbols \(\mu\) and \(\nu\), respectively. All derivatives are taken with
respect to time, \( t \), and the gravitational force is given by \( g_i \).

While the EoM as derived by Maxey and Riley has proved to be quite accurate for spheres in creeping flow, the motion of nuclei bubbles, along with many particulate flows, occur at Reynolds numbers which are greater than 1. To address these flows, researchers typically resort to semi-empirical EoMs. The semi-empirical EoMs solve equation 2.3 by substituting analytical representations for all of the forces on the sphere except for the drag force which is obtained from empirical data. Michaelides\[35\] has pointed out that this approach is theoretically precarious since the forces are not linearly independent when \( Re > 0 \). However, Michaelides points out that there are few other option for solving the EoM without numerically solving for the flow about the sphere, a task which is prohibitively expensive for most applications. While theoretically precarious, semi-empirical EoMs have proved to be quite accurately for low to moderate Reynolds number, a fact which Michaelides attributes to the empirically derived drag coefficients, \( C_D \), implicitly containing the nonlinear effects\[35\]. Another problem with the empirically derived \( C_D \) is that the coefficient is typically evaluated at steady-state conditions. Experiments have shown that transient values of \( C_D \) can be considerably different than steady-state values\[35\]. Even so, it has become common practice to use the steady-state values for the EoM of nuclei bubbles. Comparisons with experimental data show that the steady-state drag coefficients provide good for nuclei bubble trajectories\[33\]. The agreement indicates that the transient effects in \( C_D \) are relatively unimportant to the overall motion of nuclei bubbles.

Spherical particles can also experience lateral lift forces due to two mechanisms: particle rotation and fluid shear layers. Neither of these forces appear in Maxey and Riley’s formulation because lateral forces are inconsistent with creeping flow assumptions and only appear at non-zero Reynolds numbers. The lift due to particle rotation, known as the Magnus effect, is very difficult to evaluate unless the flow about the particle is being computed numerically. Because of this difficulty, this force is almost always neglected from EoMs. The lift due to a fluid shear layer, known as the Saffman effect, occurs when there is a non-zero slip velocity. While theoretical formulations for this force exist at the limit of creeping flow and inviscid flow, Sridhar and Katz have found that the actual lift force for bubbles in vortices does not precisely match the theories at either limit\[33\]. These authors derived an
empirical lift coefficient, \( C_L \), which mimics the formulation for \( C_D \). More about this empirical \( C_L \) will be provided in Chapter 3.

The EoMs discussed thus far only consider the forces on rigid spheres. However, nuclei bubbles have a dynamic radius which can increase or decrease the coefficient of drag depending on whether the radius is increasing or decreasing. Formulations for the drag correction in the limit of creeping flow are quite complex whereas drag corrections at the limit of inviscid flow are considerably simpler. Drag corrections for inviscid flow are presented in Johnson and Hsieh\textsuperscript{36} as well as the appendix of Holl and Kornhauser\textsuperscript{37}. While Johnson and Hsieh presented this term in their EoM, they did not use the term in their numerical computations of the motion of cavitation nuclei. They justified this neglect by observing that this term is only significant during a rapid change in bubble volume; an event associated with the occurrence of cavitation. Since the authors would stop computation of the nuclei bubble when it cavitated, they found that the particle trajectories were essentially unchanged when this term was neglected.

This review of the EoMs shows that nuclei bubbles have drag coefficients which are the same as solid spheres. It also shows that semi-empirical EoMs are the current best practice for economically computing the trajectories of solid spheres at non-zero Reynolds numbers. In the next section we will review the results of several studies into the dynamics of rigid bubbles in vortical flows, similar to what will be expected in a high-Reynolds number jet.

### 2.3 Numerical Studies of Rigid Bubble Motions

The semi-empirical equations of motion which were discussed in the previous section have been used by several authors to study the motion of small spherical bubbles in liquid flows. Chung and Troutt\textsuperscript{38} studied the dispersion of bubbles from an axisymmetric jet using the discrete vortex method (DVM). The authors found that the bubble response was characterized by a bubble response time, \( \gamma_\tau \), which effectively modified the drag force on the bubble in the equation,

\[
\frac{dV}{dt} = \frac{f}{\gamma_\tau} (U - V)
\]  

(2.5)
where $V$ is the velocity of the bubble, $U$ is the velocity of the fluid at the location of the bubble, and $f$ is a drag factor which is given by Clift et al.\cite{39} as,

$$f = 1 + 0.15Re_b^{2/3}. \tag{2.6}$$

The Reynolds number of the bubble is defined as,

$$Re_b = \frac{2a|U - V|}{\nu} \tag{2.7}$$

where $a$ is the radius of the spherical particle or bubble. The authors identified three response regimes depending on the chosen value of $\gamma \tau$: bubbles which exhibit the same dispersion as the fluid, bubble which exhibit less dispersion than the fluid, and bubbles which experience greater dispersion than the surrounding fluid. These three response regimes are identified by bubble response times corresponding to small $\gamma \tau$, large $\gamma \tau$, and intermediate values of $\gamma \tau$, respectively.

Wang, Troutt, and Crowe\cite{40} recognized that it is not the mass of the gas within the bubble that is important to defining the time constant, but instead the fluid virtual mass. They likened the resulting term to a modified Stokes number defined as,

$$St' = \frac{\rho U_\infty (2a)^2}{24\mu L_\infty}. \tag{2.8}$$

In this equation, $U_\infty$ and $L_\infty$ are the characteristic velocity and length scales of the liquid flow, $a$ is the radius of the bubble, $\rho$ is the density of the fluid, and $\mu$ is the dynamics viscosity of the fluid. The authors used a modified bubble drag force equivalent to $2/3$ of solid particle drag to define the modified Stokes number of a bubble. Similar to Chung and Troutt, the authors found that when $St' << 1$ the bubble follows the local fluid streamline. Values of $St' \sim 1$ resulted in bubbles which moved to the cores of vortices and values of $St' >> 1$ resulted in bubbles which are trapped in chaotic orbits inside of vortices. Research on cavitation nuclei has shown their drag coefficients to be the same as those of solid spheres. Thus equation \ref{2.8} must be modified slightly to obtain a modified Stokes number for cavitation nuclei, $St'_n$. The form of the modified Stokes number for a nuclei bubble is,

$$St'_n = \frac{\rho U_\infty (2a)^2}{36\mu L_\infty} \tag{2.9}$$
The results is that modified Stokes number for a nuclei bubble will be smaller than the modified Stokes number of a clean bubble. This tells us that nuclei bubbles will follow the local fluid velocity more closely than clean bubbles.

The studies of Chung and Troutt and Wang et al. only considered the ratio of particle inertia to particle drag. Stewart and Crowe\[41\] also considered buoyancy and found that the ratio of local liquid acceleration to gravitational acceleration is more important to the response of bubbles than the concept of the modified Stokes number. The authors recognized that the modified Stokes number will always be small for bubbles and that buoyancy can pull bubbles out of shear layers and eddies, significantly changing the response to the flow. Ruetsch and Meiburg\[42\] considered both inertia and buoyancy and found that the motion of the bubbles was characterized by an inertia parameter, $A$, and a scaled particle settling velocity, $W$. These parameters are defined as,

$$A = \frac{9}{2} \left( \frac{\rho_0 + \frac{1}{2} \rho}{\rho} \right) \left( \frac{L_\infty}{a} \right)^2 \frac{1}{Re_f},$$  \hspace{1cm} (2.10)$$

and,

$$W = \frac{2}{9} a^2 \left( \frac{\rho_0 - \rho}{\mu U_\infty} \right) g,$$  \hspace{1cm} (2.11)$$

where $\rho$ and $\rho_0$ are the densities of the fluid and gas within the bubble, respectively, and the Reynolds number is defined as, $Re = U_\infty L_\infty / \nu$. The variable $a$ is used to denote the radius of the bubbles and should not be confused with the parameter, $A$, which represents an inertia parameter and is analogous to the inverse of the modified Stokes number. The scaled particle settling velocity, $W$, is analogous to the inverse of the ratio of local liquid acceleration to the acceleration of gravity. Ruetsch and Meiburg found that the rate of entrapment by bubbles in a vortical flow is a function of $A$ only, whereas the percent of bubbles captured by a vortex increases as $W$ decreases. In other words, the modified Stokes number determines the rate at which bubbles are drawn into a vortex, but $W$ determines if gravity will overcome this pull and prevent bubbles from becoming trapped. This studied also identified 3 response regimes for bubbles and concluded that the highest concentration of bubbles at the vortex core would occur when $A$ was of order 1. When $A << 1$, the bubbles would overshoot the vortex core and enter into chaotic
orbits about the core. Bubbles corresponding to $A >> 1$ tended to follow the flow and were not drawn into the vortex cores. Thus when $A$ is less than or of order 1, the bubbles will be significantly affected by the presence of vortical structures.

Sene, Hunt, and Thomas$^{[43]}$ examined the dispersion and trapping of bubbles in a developing shear layer which was numerically modeled using a 2-d discrete-vortex method (DVM). The authors examined how $A$ (or the modified Stokes number) affected the ability of vortical structures which arise from Kelvin-Helmholtz instabilities to entrap bubbles. The authors confirmed that when the buoyancy force is too strong (large $W$) the bubbles are not effectively trapped by the vortical structures. Furthermore, the authors verified that large values of $A$ (Stokes number) indicate that bubbles will follow the local fluid velocity and will not be effectively entrapped by vortical structures. They also confirmed the observations of Ruetsch and Meiburg that bubbles corresponding to an $A$ value of order 1 are quickly drawn into and entrapped by the cores of vortical structures and that values of $A << 1$ correspond to vortical structures which trap the highest percentage of bubbles into chaotic orbits in the vortical structures.

These studies of rigid bubbles have shown that there are at least two dimensionless parameters, $A$ and $W$, which govern how a bubble will respond to a vortex in a flow. None of these authors discussed how the relative magnitude of these parameters change as the length and velocity scales of a flow vary. Both $A$ and $W$ involve the length and velocity scale of the problem and both parameters will change as the length scale of the problem is varied. The change in $A$ and $W$ with the length scale of the flow is a topic which requires further investigation. Another topic to investigate further is how much $A$ and $W$ can be altered by the growth of bubbles in response to dynamic pressures in a flowing liquid. All of the studies listed in this section have assumed that the bubbles have a fixed radius.

2.4 Numerical Studies of Cavitation Inception

The previous studies of rigid nuclei bubbles have show that small bubbles can be drawn into and trapped by vortices in a flow. The center of these vortical structures is often associated with the lowest pressure in a flow. These low pressure regions cause the trapped bubbles to grow. Under certain conditions, this growth can
lead to cavitation. The following collection of papers have examined the problem of cavitation inception by numerically modeling both the transport and radial dynamics of cavitation nuclei bubbles. The authors of these papers have tried to determine when cavitation inception is likely to occur by numerically calculating the response of nuclei bubbles to the pressure fluctuations that result from the dynamic motions of a liquid flow.

The first numerical study of cavitation inception seems to be the work of Johnson and Hsieh[36], published in 1966. These authors considered how the flow over a 2-dimensional headform affects the trajectory of cavitation nuclei bubbles. The trajectory of the bubbles was found by integrating an empirical EoM. This equation of motion was in turn a function of the bubble radius, which was determined by solving an equilibrium form of the R-P equation. This equilibrium equation, which will be derived in section 4.2.2 determines the radius of the nuclei bubble if it can be assumed that it immediately adjusts to a change in pressure with a radius which is in stable equilibrium with the background pressure. Once the bubble radius is known, the forces which accelerate the bubble can be determined from the EoM which is similar to the empirical forms discussed in section 2.2.

Johnson and Hsieh declare that a cavitation event had occurred when the bubble encounters a critical coefficient of pressure which they defined as,

\[ C_{P,\text{critical}} = -\sigma - \frac{2\sigma \left( \frac{8}{\sigma We_0} \right)^{3/2}}{3\sqrt{3} \left( 1 + \frac{8}{\sigma We_0} \right)^{1/2}} \]  \hspace{1cm} (2.12)

where \( \sigma \) is the cavitation number and \( We_0 \) is the initial Weber number of the bubble defined as,

\[ We_0 = \frac{2\rho U_\infty^2 R_0}{S}. \]  \hspace{1cm} (2.13)

The symbol \( S \) is used to define the surface tension of the liquid-gas interface which defines the bubble. The definition of \( C_{P,\text{critical}} \) is,

\[ C_{P,\text{critical}} = \frac{p^* - p_\infty}{\frac{1}{2} \rho U_\infty^2} \]  \hspace{1cm} (2.14)

where \( p^* \) is the critical pressure which leads to cavitation inception. Substituting
equation 2.14 and the definition of $\sigma$ into equation 2.12 and solving for $p^*$ reveals that this definition of the critical pressure gives $p^* \leq p_v$. This definition requires that the local pressure always be less than the vapor pressure of the liquid. This corresponds to $\sigma_i \leq -C_{P_{min}}$. Experimental data often shows that $\sigma_i > -C_{P_{min}}$, meaning that this definition of cavitation inception is not appropriate. This definition makes $C_{p,\text{critical}}$ correspond to the point where nuclei bubbles experience unstable unbounded growth, whereas cavitation is associated with unstable collapse or stable growth to a physical size which is visually detectable. So while the authors used this criteria for cavitation in their study, it does not appear to be a good choice for cavitation studies in general.

Numerically solving for the equilibrium radius and EoM allowed Johnson and Hsieh to observe how nuclei bubbles were affected by the high pressure stagnation point on the front of the headform and the low pressure recovery region over the top of the headform. The authors observed that a screening process occurred in this flow configuration where the large nuclei bubbles were pushed away from the headform by the high pressure stagnation region. The screening process prevents the largest, most unstable bubbles from experiencing the minimum pressure which occurs on the surface of the headform in the recovery region. Prior to this study by Johnson and Hsieh, it was commonly believed that cavitation was purely a function of the stability of the large bubbles in a flow. Johnson and Hsieh showed that the history of the nuclei bubbles can play an important part in the inception process by selectively screening out the most unstable nuclei bubbles. This work is significant for demonstrating that models which do not account for the history of bubbles may miss important physics which are important to determining $\sigma_i$ for a flow.

The next numerical study of cavitation inception which is relevant to the present work is a paper by Latorre which builds upon the methods of Johnson and Hsieh by using the full R-P equation to determine the volumetric change of cavitation nuclei bubbles. Latorre attempted to examine how the tip vortex from a propeller affects the cavitation nuclei bubbles. The author used a Rankine vortex as a model for the propeller’s tip vortex and improved upon Johnson and Hsieh’s cavitation inception definition by calculating the acoustic emissions of a spherical bubble using the model of Fitzpatrick and Strasberg and calling cavitation
when the acoustic emissions reached a critical value. The results of these numerical methods were reported to be in good agreement with the characteristic frequency of sound bursts observed in experiments and allowed the author to numerically determine the incipient value of the cavitation number, $\sigma_i$. Latorre’s research shows that the combination of an empirical EoM and Rayleigh-Plesset bubble dynamics has the potential to allow a priori determination of incipient values of cavitation inception. Although the author does not discuss the matter, it is interesting to point out that both the experimental and numerical data appear to show size effects on $\sigma_i$. In both sets of data, $\sigma_i$ increases with length scale; a trend that is similar to the scale effects observed in jet cavitation.

In the early 1990’s Meyer et al. [10] returned to the problem of a headform. These authors used a semi-empirical EoM and the full Rayleigh-Plesset equation to evaluate the response of nuclei bubbles to the flow over a 2-dimensional Schiebe headform. The flow field was determined by solving the steady-state 2-dimensional Reynolds-Averaged Navier-Stokes (RANS) equations for the Schiebe headform geometry. The cavitation nuclei bubbles were randomly released into the flow along the $C_p = 0$ line which lies between the high pressure stagnation region on the nose and the low pressure recovery region over the top of the headform. The radius of the bubbles were varied so that the distribution of all released bubbles would match the distribution of naturally occurring bubbles in the experimental test facilities at the Garfield Thomas Water Tunnel. By releasing bubbles on the $C_p = 0$ line, the authors effectively by-passed the screening mechanism of headforms observed by Johnson and Hsieh. Thus the cavitation number in these experiments was a function of the most unstable nuclei bubbles in the flow.

Meyer et al. conclude from their study that traveling bubble cavitation is very sensitive to nuclei size. They also conclude that the minimum on-body $C_p$ is not useful for determining incipient cavitation values because the cavitation nuclei never reach the surface. The authors propose an impulse function which they believe to be important for cavitation scale effects. The function represents the total time which a nuclei bubble is exposed to a pressure which lies below the vapor pressure for the liquid. As the length scale of a dynamically-similar flow increases, so does the length of time which a bubble is exposed to the low pressure region. This will give the bubbles more time to grow and makes them more likely
to be observed at higher cavitation numbers. Since the impulse function measures the time which the bubble is below the vapor pressure, the theory implies that the minimum pressure in the flow must be below the vapor pressure. This restricts $\sigma_i \leq -C_{P_{\text{min}}}$, where $C_{P_{\text{min}}}$ is the minimum pressure coefficient in the vicinity of cavitation. However, data from many cavitation inception studies\cite{2,37} show that $\sigma_i$ is often larger than $-C_{P_{\text{min}}}$, indicating that cavitation inception occurs when nuclei bubbles are exposed to pressures which are higher than the vapor pressure of the fluid. While this does not prove that time effects are not important, it does show that the criteria in the definition of impulse function is not sufficient.

As computational power increased through the 1990’s, it became possible to numerically solve the steady-state RANS equations for complex 3-dimensional geometries. Hsiao and Pauley\cite{46} took advantage of increased computational power to model cavitation inception in the tip vortex of a finite-span hydrofoil. The results of this study showed that the nuclei bubbles, which were governed by the R-P equation and a semi-empirical EoM, needed to pass through a “window of opportunity” to be drawn into the tip vortex and cavitate. The window of opportunity, first proposed by Maines and Arndt\cite{47}, is a region of space in front of the hydrofoil through which nuclei bubbles must pass in order to reach the low pressure tip vortex. Hsiao and Pauley observed that the window of opportunity becomes larger as the nuclei bubble radius increases, a consequence of increased sensitivity to pressure gradients as nuclei radii increase. The measurement of the window of opportunity is essentially another way to measure the ability of the tip vortex to capture the nuclei bubbles, which is analogous to the $A$ parameter defined by Ruetsch and Meiburg\cite{42} or the modified Stokes number defined by Wang et al.\cite{40}. Unlike Johnson and Hsieh which showed that larger bubbles are screened from the occurrence of low pressure, the larger bubbles are actually drawn into the tip vortex from a large window of opportunity. This ensures that the largest and most unstable bubbles which pass through this window will be responsible for cavitation inception in separated flows governed by strong vortical structures.

The focus of the present study is on cavitation inception in high-Reynolds number circular jets; flows which differ considerably from headforms and tip vortices. A previous simulation of cavitation inception in jet flows was conducted by Cerutti, Knio, and Katz\cite{21}. These authors focused on the effect that jet slenderness ratio,
defined as the ratio of the shear layer thickness and the jet diameter, and the effect that the jet Reynolds number have on $\sigma_i$. The jet flow was modeled by solving the axisymmetric streamfunction vorticity equations with no turbulence model. This method is essentially an implicit large eddy simulation (ILES) where no subgrid dissipation is added to the flow to account for turbulence.

Cerutti et al. neglected all changes in bubble radius and determined that a bubble “cavitated” when it was exposed to a pressure that was equal to or less than a critical pressure; a similar criteria to that used by Johnson and Hsieh. As discussed with Johnson and Hsieh, this criteria leads to a situation where the minimum pressure in the vicinity of cavitation must be below the vapor pressure of the fluid. The trajectory was calculated using the Sridhar and Katz\[33] EoM which includes an empirical coefficient of lift which was briefly mentioned in section 2.2. Using these methods, the authors showed that the minimum $C_p$ within a jet flow will decrease as the exiting shear layer thickness decreases (i.e. stronger shear layers) and the $x/D_J$ location where cavitation inception occurs will also decrease as the shear layer decreases (i.e. cavitation will occur closer to the jet as the shear layer strength increases).

The study of Cerutti et al. also confirmed that larger bubbles are more easily entrained into the vortical structures of the flow, confirming the conclusions of the studies in section 2.3. The authors also observed that the entrainment of small nuclei bubbles (on the order of 10µm) was strongly dependent on Reynolds number, whereas the entrainment of larger bubbles, which are more unstable, was found to be insensitive to Reynolds number. The authors hypothesize that the experimentally observed trend of increasing cavitation number with increasing jet size is a consequence of the inability to maintain similarity of shear layer thickness between model scale and full scale. They propose that as $D_J$ increases, the ratio of the shear layer thickness to the jet radius decreases, effectively causing larger jets to have a stronger shear layer and thus lower minimum values of $C_p$.

All of the previous studies of cavitation inception, and most of the studies for single bubbles subjected to acoustic forcing, assume that the bubbles remain spherical at all times. Hsiao and Chahine\[48] noted that nuclei bubbles were observed to distort in experimental studies of tip vortex flows. They developed a non-spherical bubble model based on kinematic free surface boundary conditions. The authors
compared the equivalent radius of the non-spherical model with the bubble radius predicted by the standard R-P equation and an improved R-P equation which they called the SAP model (surface-averaged pressure). The SAP model averages the pressure at 6 polar points on the bubble surface instead of using the pressure at the center of the bubble. This method is recommended for flows where the pressure gradients are significant on a length scale proportional to the bubble radius. The authors found that for tip vortex flows, the traditional R-P equation would over-predict the size of the bubble radius because the external pressure was taken to be the pressure at the center of the bubble. This pressure would be the vortex core pressure and not the pressure at the surface of the bubble, which tends to increase as the bubble expands further out into the vortex. The SAP model also includes a pressure correction term which is derived by integrating the pressure over the surface of a sphere in the limit of inviscid flow. This is a questionable modification. Experimental evidence indicates that bubbles behave as a solid sphere and typically have slip velocities which correspond to low Reynolds number conditions. Regardless, the authors found that the SAP radius and non-spherical equivalent radius were nearly identical for tip vortex flows. Since the non-spherical model is considerably more expensive to implement, the authors concluded that the SAP model would be sufficient for studies of cavitation inception in tip vortex flows.

These authors also examined two different definitions for cavitation inception: visual and acoustic. They found that the visual and acoustic criteria for declaring cavitation inception resulted in significantly different values of \( \sigma_i \). This is attributed to the different physics which govern the observations. Visually detected cavitation requires that cavitation nuclei grow to a radius which can be repeatably observed in a transient flow (often reported to be a diameter of 1 mm). Thus a large stable cavity is the most likely to be observed and very short duration unstable cavities are likely to be missed. Acoustic detection relies on unstable cavities which undergo a series of violent collapses and rebounds which radiate acoustic energy. Thus the physical size of the nuclei bubble is less important in acoustic measurements of cavitation inception. The authors conclude that the modeled definition of cavitation inception should reflect the same physics which are used to experimentally detect cavitation inception, otherwise large discrepancies between numerical models and experiments are likely.
Hsiao, Chahine, and Liu\textsuperscript{11} used the SAP model to further examine bubble dynamics in tip vortex flows and found that the most significant acoustic emissions from nuclei bubbles occurs when bubbles undergo rapid growth or a violent collapse. These authors investigated the violent collapse events and found that they corresponded to bubbles which have grown in response to a local low pressure region and which then advect into an adverse pressure gradient. The enlarged bubble becomes unstable under the increasing pressure, rapidly decreasing in size until the internal pressure builds to a point which reverses the rapid contraction. The authors noted that while previous studies have made great progress in understanding bubble dynamics and cavitation inception, they have not done a good job of accurately predicting incipient cavitation numbers for general flows. They attribute this failure to inadequate models of the flow field which do not accurately capture the transient low pressure events and not as a failure of the cavitation inception models. The authors believe that accurate transient predictions of the flow field are critical to obtaining accurate predictions of $\sigma_i$.

To address the inability of computational fluid dynamics (CFD) methods to accurately model the transient pressure fluctuations, Farrell\textsuperscript{49} returned to the 2-D Schiebe headform and applied a correction to the pressure field which accounts for the unresolved pressure fluctuations. The magnitude of the correction was set by the local turbulent kinetic energy (TKE) as suggested by Hinze\textsuperscript{50}. The TKE was obtained from the CFD turbulence model. Cavitation nuclei were released upstream of the Schiebe headform and were modeled using the R-P equation and the EoM from Meyer et al.\textsuperscript{10}. The authors found that the corrections to the pressure and velocity field resulted in a negligible effect on the value of $\sigma_i$. However, the review of nonlinear bubble dynamics in section 2.1 revealed that bubble responses are frequency dependent, an effect which is completely neglected by the pressure correction applied by Farrell.

Kim, Paterson, and Stern\textsuperscript{51} recognized the frequency dependence of nuclei bubble responses and implemented a transient turbulent forcing to the problem of cavitation inception in a ducted marine propulsor. These authors modeled the ducted propulsor using a 3-D overset grid and steady-state RANS CFD methods. Cavitation nuclei were modeled using the R-P equation and an empirical EoM with an acoustic definition of cavitation inception. The authors performed
a thorough verification and validation with comparison to experimental data and found that these CFD simulations accurately predicted the hydrodynamic forces on the propulsor. They implemented a transient subgrid pressure forcing which consisted of a small amplitude sinusoidal perturbation superimposed on the local pressure. The frequency of the forcing was chosen to correspond to the frequency of Kolmogorov-scale turbulent fluctuations which were well below the linear resonance frequency of the nuclei bubbles. The authors found that this subgrid forcing had almost no effect on the resulting cavitation number in their simulation. They also found that the numerical simulations under-predicted the experimentally-obtained incipient value of the cavitation number by 20%. They concluded that this under-prediction was a result of an under-resolved tip vortex and suggested that the problem required better grid resolution and better turbulence models.

As already mentioned, the review of nonlinear bubble dynamics in section 2.1 suggests that the frequency sensitivity of bubbles is highly nonlinear and that bubbles can be receptive to frequencies much lower than the linear resonance frequency. Thus, it is possible that the choice of Kim et al. to use the Kolmogorov frequency as the subgrid pressure frequency may have been too high of a frequency, instead of too low as the authors suggested. The lower frequency turbulent fluctuations in the flow contain more kinetic energy than the Kolmogorov-scale velocity fluctuations. As such, these lower-frequency fluctuations can result in lower pressures than the higher-frequency fluctuations. It will be important to consider the frequencies to which the nonlinear bubbles are receptive, especially near the point where the bubbles begin to cavitate.

These research papers regarding numerical simulation of cavitation inception have provided important information about issues which must be considered in the numerical simulation of cavitation inception in jets. These studies have shown that meaningful comparisons with experimental data can only be made if the experimental and numerical definition of cavitation inception are compatible. They have also shown that the combination of the R-P equation and semi-empirical EoM has the potential to accurately predict \( \sigma_i \). However, the pressure field produced by the liquid flow must be accurately represented by the numerical simulation. To ensure that the numerical simulation represents a physical pressure field, the numerical results need to be compared with experimental data. The following
section reviews the literature for high Reynolds number circular jets. The data from this review will be used to ensure that the pressure field in the present numerical simulation of a jet flow is physically realistic.

2.5 Circular Jet Dynamics

The statistical nature of high-Reynolds number circular jets gives the impression that the jet flow is an orderly, well-defined flow. Rouse published a series of papers\textsuperscript{[4, 6, 52]} between 1950 and 1966 in which the characteristics of the statistics of jet flows and the nature of cavitation inception in the near field of circular jets was reviewed. In these reports, Rouse discusses two distinct regions of the flow: the zone of flow establishment and the zone of established flow, illustrated in the upper half of figure\textsuperscript{[2.1]}. The zone of established flow is characterized by the existence of a central “irrotational” core from $x/D = 0$ to $x/D \sim 6$. The irrotational core has an invariant centerline mean velocity throughout the zone of flow establishment and this region is separated from the surrounding irrotational fluid by an expanding shear layer. The shear layer is expanding inward as well as outward and converges to the centerline of the jet at the end of the zone of flow establishment. One point to note is that Rouse declares the zone of flow establishment to end at $x/D_J = 6$, but his data indicates that the actual location should be somewhere between $4 < x/D_J < 6$.

The zone of established flow consists of the region downstream of $x/D_J \sim 6$. This region is characterized by a centerline mean velocity that decays with $x/D_J$ and a mean velocity profile that expands radially with increasing $x/D_J$. Rouse reports that the rate of spreading is $SR = 1/5$ (1 unit radially for every 5 units downstream). The figure in which Rouse reports this value only extends to $x/D_J < 15$, and it is not entirely clear just how much of a constant this value really is since other data\textsuperscript{[53]} indicates that the spreading rate is $SR \sim 0.1$ in the self-similar region of the jet at $x/D_J > 30$.

Rouse also reported values of the normalized root-mean-square (RMS) of the
Figure 2.1: Expected statistical representation of a jet (upper half) and instantaneous representation (lower half) of a high-Reynolds number circular jet. The statistics are adapted from Rouse\textsuperscript{4} and the unsteady dynamics are adapted from Yule\textsuperscript{5}.

Velocity fluctuations\textsuperscript{52} defined as,

$$\frac{\sqrt{u'^2}}{U_J},$$

and the pressure fluctuations defined as,

$$\frac{\sqrt{p'^2}}{\frac{1}{2}\rho U_J^2}.$$  (2.16)

The maximum normalized RMS velocity fluctuations were found to be on the order of 0.25 along the centerline of the shear layer from $0 < x/D_J < 3$ while the maximum normalized RMS of the pressure fluctuations was reported to be on the order of 0.18 from $0 < x/D_J < 3$. The pressure fluctuations were measured using an invasive static pressure probe with a cylindrical piezoelectric crystal.

While the statistical nature of the jet flow is orderly and well-defined flow, the transient dynamics of the jet flow exhibit rich dynamics which appear to be highly sensitive to the exit conditions of the upstream nozzle flow. The review articles of E. J. List\textsuperscript{54} and Grinstein et al.\textsuperscript{55} provide good overviews of the unsteady dynamics which can expected from a high-Reynolds number circular jet. The basic
sequence for the flow is as follows:

1. The high-speed jet flow causes a thin shear layer to be produced with a thickness, \( \delta_0 \).

2. This shear layer experiences exponential growth of a Kelvin-Helmholtz instability which causes the shear layer to roll-up into a series of vortex rings.

3. The vortex rings force the exchange of "turbulent" jet fluid and irrotational ambient fluid.

4. The motion induced in the fluid by each vortex imparts an influence on neighboring vortices in such a way as to cause adjacent vortices to pair-off and eventually coalesce.

5. The vortex rings develop a secondary circumferential instability that causes the vortex rings to eventually breakup.

All of the dynamics described in this list occur in the zone of flow establishment, as illustrated in figure 2.1.

The first distinctive structure of a jet flow is the thin shear layer created when the flow exits the nozzle. In many experimental studies, this shear layer is laminar and quite thin, often on the order of \( \delta_0 = (1/100)D_J \). Several authors have noted that the formation and interaction of the vortex rings in the jet flow have a strong dependence on the jet slenderness ratio, \( \delta_0/D_J \). Cerutti, Knio, and Katz[21] numerically examined the effects of the jet slenderness ratio on the dynamics of a jet flow with Reynolds numbers between \( 2 \times 10^4 \leq Re \leq 2 \times 10^5 \) and discovered that the number of vortices formed by the instability of the shear layer increases as \( \delta_0/D_J \) decreases. The authors also found that the jet Strouhal number based upon the shear layer thickness \( (St_{\delta_0} = f \delta_0/U_J) \) remained relatively constant, with a value of 0.1, while the Strouhal number based upon jet diameter \( (St_D = f D_J/U_J) \) was found to decrease as \( \delta_0 \) increased. The decrease in \( St_D \) was in response to the decrease in the number of vortex rings which form when \( \delta_0 \) increases.

Another important characteristic of the shear layer is its turbulent state. Most experimental studies of turbulent jet flows are created by, or at least are assumed to be created by, laminar shear layers exiting from a nozzle or orifice. Gopalan,
Katz, and Knio [20] looked at how a turbulent shear layer, formed by tripping the boundary layer within the nozzle, would change the dynamics of the jet flow. The authors tested a circular jet at $Re = 5 \times 10^5$ and found that the thin laminar shear layers quickly transitioned to a 3-dimensional flow characterized by coherent hairpin-like vortices upstream of the Kelvin-Helmholtz instability. The turbulent shear layers, on the other hand, did not exhibit strong coherent streamwise flow structures and only experienced the Kelvin-Helmholtz instability. While both flows eventually encounter the Kelvin-Helmholtz instability, the existence of the strong streamwise hairpin vortices had important implications for cavitation inception. Cavitation in the laminar shear layers was found to occur within the hairpin vortices at $x/D_J < 0.6$, prior to inception further downstream. The results is that the thin laminar shear layer experiences cavitation at $\sigma_i = 2.5$ which is a higher cavitation number than the turbulent shear layer jet which was found to cavitate at $\sigma_i = 1.7$.

Regardless whether the shear layer was laminar or turbulent, it eventually became unstable and rolled-up into a series of vortex rings. The location of the roll-up and the spacing of the vortex rings was a function of the Kelvin-Helmholtz instability of the shear layer. Gopalan et al. [20] observed that coherent vortex rings appeared to form at $x/D_J = 0.75$ for the laminar shear layer and in the neighborhood of $x/D_J \sim 0.3$ for the turbulent boundary layer. No mention is made regarding the first vortex pairing and coalescence for the laminar shear layer case, but the turbulent shear layer case was reported to undergo vortex pairing and coalescence at $x/D_J \sim 0.6$. The numerical simulation of a 2-dimensional axisymmetric jet presented by Grinstein et al. [55] showed that vortex rings form in the range $0.4 < x/D_J < 0.8$. The initial shear layer was laminar and very thin in these simulations with $\delta_0/D_J = 1/140$ and the Reynolds number was $Re = 1.5 \times 10^5$. As Gopalan et al. [20] noted, the thin laminar shear layer should quickly develop 3-dimensional structures which the 2-dimensional numerical simulations cannot resolve. As such, the numerical simulations only capture the 2-dimensional Kelvin-Helmholtz instability, creating uncertainty as to the prediction of where the vortex rings first form.

As the vortex rings form, they induce motions on the other vortex rings. These induced motions tend to cause the vortex rings to pair-off with one or two neigh-
boring vortex rings. As already pointed out, vortex pairing occurs quickly with the first coalescence of vortex rings reported as close as $x/D_J \sim 0.6$ by Gopalan et al.\cite{20}. The simulations of Grinstein et al.\cite{55} indicate that the first vortex coalescence occurs in the range $1.0 < x/D_J < 1.85$. The data of Grinstein et al. and experimental data from Ran and Katz\cite{14} illustrate the random variations discussed by Yule\cite{5}. These authors observed that the total number of vortex ring pairing and coalescing events varied between 2 and 3 events. Experiments by Ran and Katz\cite{14} indicate that the second vortex pairing occurs in the range of $1.5 < x/D_J < 3.5$, a range which is supported by simulations of Grinstein et al. which showed the second vortex pairing to occur between $1.8 < x/D_J < 2.3$ and the third pairing to occur in the range $3.6 < x/D_J < 5.2$.

The local Strouhal number of the jet decreases with $x/D_J$. This occurs as the vortex ring spacing increases with vortex rings coalesce, causing the frequency associated with the dominant fluctuations in the flow quantities to decrease. The dominant Strouhal number for high-Reynolds number jet flows was reported by List\cite{54} to lie within the range $0.3 \leq St_D \leq 0.5$. The advection speed of the vortex rings (presumably after the second vortex pairing) has been reported by Lau and Fisher\cite{56} to be on the order of $0.65U_J$ with a vortex ring spacing of $1\frac{1}{4}D_J$. The authors however admitted that while the vortex ring speeds remained fairly consistent, there was a considerable amount of variation to the vortex spacing. Furthermore, Cerutti et al. have already shown that the number and spacing of the vortex rings is highly dependent upon the shear layer thickness. The axisymmetric numerical simulations of Grinstein et al. also show considerable variation in the spacing between vortex rings but provide an average spacing which is close to $1\frac{1}{4}$. It is important to note that Lau and Fisher did not report $\delta_0/D_J$ ratios for their experimental jets. These authors did, however, allude to the fact that they have a laminar exit shear layer.

The final destruction of the coherent vortex rings toward the end of the zone of flow establishment is generally attributed to a circumferential instability which develops in the vortex rings. The mechanism for this instability is explained and demonstrated both experimentally and numerically by Meiburg and Lasheras\cite{57}, albeit for a plane wake. In a later paper, Martin, Meiburg, and Lasheras\cite{58} show that the mechanism for the formation of streamwise vorticity and circumferential
perturbations in an axisymmetric jet wake is quite similar to the mechanism in plane wakes, albeit the plane wake lacks vortex stretching mechanisms which are present in the axisymmetric jet wake. The instability arises due to the stretching of vorticity filaments which lie between two neighboring vortex rings. This vorticity is initially weak, but as the vortex filaments are stretched around the adjacent vortex rings, the vorticity of the filaments increases until they are strong enough to induce a significant perturbation of the vortex rings.

How these dynamics affect the pressure field, and ultimately the inception of cavitation, will be a key question which must be addressed in the present work. Gopalan et al.\cite{20} have already demonstrated that the state of the exiting shear layer is very important, not only for the dynamics of the jet, but also the location of cavitation inception and the value of $\sigma_i$. They found that a thin laminar shear layer leads to cavitation inception in the hairpin-like vortices which form in the region $x/D_J < 0.6$ and resulted in $\sigma_i = 2.5$. The turbulent shear layer was found to induce cavitation at the location of vortex pairing and coalescence at $x/D_J \sim 2.0$ with $\sigma_i = 1.7$. It is interesting to note that both of these cavitation numbers are larger than the value discovered in an earlier study by Ran and Katz\cite{14} which used a smaller nozzle (2.5 cm versus 5.0 cm). The smaller nozzle which was operated at the same Reynolds number ($4.5 \times 10^5$) resulted in a smaller incipient cavitation number, $\sigma_i = 1.0$, and showed little sensitivity to the turbulent state of the boundary layer inside of the nozzle. The insensitivity to the turbulent state is in opposition to the sensitivity observed by Gopalan et al.

The numerical studies of Cerutti et al.\cite{21} indicate that the cavitation inception number will be sensitive to the jet slenderness ratio. Their numerical studies found that lower minimum pressures will occur as the slenderness ratio is decreased (i.e. the shear layer becomes thinner). The authors concluded that jets with thinner shear layers would be more susceptible to cavitation and therefore would have a higher $\sigma_i$. Trends with Reynolds number are quite interesting. Ran and Katz\cite{14} present results which showed that $\sigma_i$ increased as Reynolds number decreased. However, the numerical simulations of Cerutti et al. indicate that $\sigma_i$ has little dependence upon Reynolds number. The key to explaining these differences may be the jet slenderness ratio. In the numerical experiments, Cerutti et al. were able to modify the Reynolds number of the flow while maintaining the thickness of the
shear layer exiting the jet. Ran and Katz provide no mention of slenderness ratio or shear layer state in the two experiments, leaving open the possibility that the jet slenderness ratio changed with Reynolds number. Cerutti et al. hypothesize that much of the confusion regarding cavitation scale effects in jets and conflicting trends with jet Reynolds number can be blamed on the lack of control and monitoring of the slenderness ratio in experiments. Very few experimental studies of jet flow provide adequate details regarding the jet shear layer thickness and turbulent state of the flow exiting the nozzle.

In studying the pressure fluctuations and resulting cavitation inception in circular jets of $Re \sim 500,000$, Ran and Katz[14] found the minimum coefficient of pressure was -0.97 at $x/D_J = 2.5$. This value was measured by examining the response of bubbles to the pressure in the liquid flow. The location $x/D_J = 2.5$ corresponds to the region of the second vortex pairing and coalescence. This suggests that vortex pairing is responsible for the lowest pressures in the zone of flow establishment. The authors also looked at pressure fluctuations further downstream in the zone of established flow since many cavitation studies of jet flows report visual inception in this region. The authors found that the minimum coefficient of pressure in the zone of established flow is on the order of -0.6, significantly higher than the minimum pressure due to vortex pairing. It is curious why much of the literature concerning cavitation inception in high-Reynolds number circular jets reports cavitation to occur in the zone of established flow and not the zone of flow establishment. These authors also pointed out that the RMS of pressure fluctuations was an order of magnitude smaller than the absolute value of the minimum coefficient of pressure in the flow. This large discrepancy indicates that RMS values of pressure fluctuations will provide little help in understanding trends in $\sigma_i$ for jet flows.

This review of cavitation inception in high-Reynolds number circular jets shows that the dynamics of the jet flow are highly dependent on both $\delta_0$ and the turbulence in the exiting shear layer. There does appear to be a significant scale effect in the results, with the larger diameter jets experiencing higher $\sigma_i$ than smaller jets. Regardless of the state of the exiting shear layer, strong vortex dynamics are reported in all of the relevant high-Reynolds number incompressible flows. It has also been demonstrated that the minimum pressure in the flow occurs because
of the interaction of the coherent vortex rings. It is still puzzling that cavitation inception is often observed in the zone of established flow where the minimum pressure is significantly higher than the pressures which are expected from vortex ring interactions in the zone of flow establishment.

### 2.6 Turbulence Modeling

The computational costs associated with a direct numerical simulation (DNS) or large-eddy simulation (LES) of high-Reynolds number flows are prohibitively large, especially when near-wall eddies are resolved. Traditionally, this has left practitioners of CFD with only unsteady Reynolds-Averaged Navier-Stokes (URANS) methods to solve engineering problems. However, in the late 1990’s a new hybrid method of turbulence modeling was proposed which allowed for resolution of the largest turbulent eddies within the flow while minimizing the costs associated with resolving near-wall turbulence\[59\]. This method has become known as detached-eddy simulation (DES) and is a hybrid between the LES and URANS methods. The DES method consists of using a traditional RANS turbulence model to approximate the turbulent eddy viscosity in the near-wall region of the flow; a region where RANS turbulence models perform well. The RANS turbulence model is then used as a sub-grid eddy viscosity model in the regions away from the wall where the grid is fine enough to perform a LES.

The original DES implementation was developed to use the Spalart-Almaras (S-A) one-equation turbulence model. This turbulence model worked well because the turbulent diffusion length appears explicitly in the formulation. This turbulent diffusion length, given by the variable \( d_w \), was replaced with a modified length scale, \( \tilde{l} \), defined as,

\[
\tilde{l} = \min(d_w, C_{DES}\Delta)
\]  

where \( \Delta \) is the maximum dimension of the local grid cell and \( C_{DES} \) is an adjustable model constant. In the S-A turbulence model, the turbulent eddy viscosity is proportional to the local deformation rate and the square of the turbulent diffusion length (i.e. \( \nu_t \propto Sd_w^2 \))[59]. By replacing the turbulent diffusion length with \( \tilde{l} \), the DES turbulence model reduces the amount of modeled TKE in regions where the
grid is fine enough to resolve large energy-containing eddies.

While a one-equation turbulence model is usually simpler to solve than a two-equation model, the latter have found wider use for general CFD computations\[60\]. The DES method was modified by Strelets\[61\] to allow for the use of the two-equation blended $k - \epsilon/k - \omega$ turbulence model of Menter\[62\] which is implemented in CFDSHIP; the code which will be used for the present work. The modification proposed by Strelets is to replace $d_w$ in equation \[2.17\] with the turbulent length from the 2-equation model. The two-equation turbulent length is not the distance to the nearest wall, as in the S-A model, but is instead a function of the turbulent kinetic energy, $k$, and the specific dissipation rate, $\omega$. Strelets also showed that the DES solutions were sensitive to the numerical scheme used and suggested using a fourth-order central difference instead of an upwind difference in the LES regions. Since many URANS codes are only second-order accurate, Strelets introduced a blending function which smoothly transitions the numerical scheme from second-order upwind differencing in the RANS regions to fourth-order central-differencing in the LES regions of the flow.

A search of the literature returned no examples of DES being used to simulate the flow of incompressible circular jets. There were, however, a few examples of DES being applied to high Mach number jet flows\[63, 64, 65\]. All of these studies were aimed at obtaining predictions of jet noise and did not make a detailed comparison with experimental data for jet flow dynamics. Therefore, little can be said regarding the ability of DES to accurately predict the hydrodynamics of jet flows. Presumably, however, if DES does a good job of reproducing the acoustic signature of a jet, it is also doing a good job of reproducing the fluid dynamics of the jet, but this has not been proved. The study by Shur, Spalart, and Strelets showed that the jet shear layer dynamics are strongly affected by the numerical methods used by the CFD code. Furthermore, these authors found that performing an LES without the subgrid turbulence model (to be referred to as an “implicit LES”, or ILES, in this work) provided the most accurate prediction of jet acoustics. An ILES relies only on numerical dissipation to account for subgrid turbulent dissipation. It is interesting that a non-physical method of accounting for the subgrid turbulent dissipation should provide the best comparison with experimental data. It is unclear if the success of ILES is purely coincidental, or
whether this method actually provides better results for general flows.

An area of concern for the DES method is the continuity of turbulent kinetic energy (TKE) in the so-called grey regions. The grey regions are the areas where the flow transitions from RANS to LES. This transition is controlled by grid spacing and the turbulent dissipation length. As the fluid flows from a RANS region into an LES region, the subgrid TKE of the flow is reduced by replacing the turbulent dissipation length with a modified turbulent dissipation length. As illustrated in figure 2.2, the resolved TKE does not immediately increase to replace the modeled TKE. The DES algorithm assumes that resolved TKE will rapidly develop in response to flow instabilities within the grey region, but there is nothing within the algorithm to ensure that this occurs. Paterson and Peltier demonstrated that the drop in TKE can lead to premature separation of detached wakes for some flow geometries. They modeled flow separating from beveled trailing edges and designed the grid such that the grey region occurred in a boundary layer flow upstream of the separation point. The boundary surface apparently maintained sufficient stability in the flow that the resolved TKE did not sufficiently develop to replace the reduced subgrid TKE. The reduced total TKE resulted in premature separation of the flow at the beveled trailing edges. These results show that the grey region can lead to errors in the numerical solution. Ultimately, it is the responsibility of the user to ensure that the grey region is not negatively affecting the results.
Figure 2.2: Illustration of the grey region in a DES flow. The grey region comprises
the space where the DES model transitions from its RANS mode (coarse grid) to
its LES mode (fine grid).
Chapter 3

Dimensional Analysis and Cavitation Scaling

3.1 Dimensional Analysis

Before completing a detailed analysis of the problem of cavitation inception, it is helpful to consider the problem from the perspective of a dimensional analysis. This process of performing a dimensional analysis allows one to gain insight into the important parameters of the problem without necessarily knowing the governing equations of the problem. In a dimensional analysis, several dimensional variables are grouped together to form a reduced set of dimensionless variables. These dimensionless variables represent the universal, or natural, variables which govern the problem. However, performing a dimensional analysis requires strong physical insight into the problem and an understanding of what physical variables are significant to the system. Omitting important physical variables from the analysis can lead to an incomplete set of dimensionless variables.

The goal of cavitation inception studies is to identify the value of the reference pressure, $p_\infty$, which will lead to cavitation inception in a flow. This reference pressure is typically the far-field pressure as it is both easy to measure and remains relatively constant. The local pressure where incipient cavitation occurs is lower than the reference pressure because of the dynamic actions of the liquid flow. The local pressure will depend on several physical variables of the flow which
Figure 3.1: Sketch of a quiescent jet and the parameters which are expected to influence the pressure at a given point within the flow.

are illustrated in figure 3.1. Dynamic motions in incompressible liquids (with no cavitation) are insensitive to the magnitude of the pressure and instead respond to the gradients of the pressure. Changing the magnitude of the reference pressure will not change the dynamic motions of the liquid, but it will change the magnitude of the minimum pressure by an equivalent amount. It is the difference between the background pressure and the local pressure within the flow that is a function of the physical variables which define the flow. The relationship to these physical variables is,

\[ p_\infty - p = f(U_J, D_J, \rho, \mu) \]  (3.1)

where \( U_J \) is the characteristic jet velocity, \( D_J \) is the diameter of the jet, \( \rho \) is the liquid density, \( \mu \) is the liquid viscosity, and \( \vec{x} \) is the spatial location where inception is noted. Experience shows that changing the reference pressure will not change the dynamic motions of the liquid until cavitation is formed. Above a certain critical value of the reference pressure, the liquid flow is independent of the reference pressure.

The difference between the reference pressure and the minimum pressure is not affected by the changes in the background pressure. This means that as the background pressure is changed, the minimum pressure in the flow will change by the same amount. The goal of cavitation studies is to determine the value of the reference pressure which leads to cavitation inception. This value is labeled as \( p_{\infty,i} \). Cavitation inception is expected to occur when the value of the minimum pressure reaches the vapor pressure (\( p_v \)) of the liquid. Since the quantity \( p_\infty - p \) in equation 3.1 is expected to remain constant as the physical value of \( p_\infty \) is changed,
the pressure difference at cavitation inception will be the same (i.e. \( p_{\infty,i} - p_v = p_{\infty} - p_{\text{min}} \)). Using the fact that the pressure difference will remain constant and equation 3.1, the difference between the reference pressure and the vapor pressure at cavitation inception can be written as a function of,

\[
p_{\infty,i} - p_v = f(U_J, D_J, \rho, \mu)
\] (3.2)

Equation 3.2 shows that the difference between the critical reference pressure and the vapor pressure is governed by the dynamic actions of the liquid. As the variables which govern the dynamic actions of the liquid change, the value of the critical pressure which leads to cavitation inception is also expected to change.

Figure 3.2: Sketch of the parameters which are expected to influence the radius of a nuclei bubble.

Research into cavitation inception has shown that it typically results from the growth of microscopic nuclei bubbles which respond to the pressures within the flow[9, 22]. Now imagine an experiment where an observer is carefully watching a flow and looking for evidence that cavitation is occurring. If the observer is watching the flow with an unaided eye, they will be able to observe a nuclei bubble when it reaches a certain size, \( A \). The same observer then watches a high-speed video of the flow which is slowed and magnified. With the aid of the video, the observer can detect nuclei bubbles of size \( B \), where \( B < A \). It is apparent that the definition of cavitation inception (i.e. the size of a detectable nuclei bubble), will be important to the magnitude of \( p_{\infty,i} \). The size of a detectable nuclei bubble, \( a_i \), can be added to the list of variables which affect \( p_{\infty,i} \),

\[
p_{\infty,i} - p_v = f(U_J, D_J, \rho, \mu, a_i)
\] (3.3)

Since \( p_{\infty,i} \) is a function of the size of a nuclei bubble, we expect that the
physical variables which determine the radial size of nuclei bubbles will also be important. These physical variables, illustrated in figure 3.2, are expected to include surface tension \( S \), the pressure outside of the bubble \( p \), and the pressure inside the bubble \( p_{in} \). The pressure outside of the bubble is a function of the variables in equation 3.1. Technically speaking, both \( S \) and \( p_{in} \) will be functions of temperature. However, most cavitation problems are assumed to be isothermal and almost all are performed at or near room temperature. Therefore, temperature effects will be neglected from this study. Compressibility of the surrounding liquid will also be neglected as the bubble wall velocity is expected to be much lower than the speed of sound for all responses except a violent collapse which occurs within a very short time span relative to the nuclei life. Several authors have examined this assumption and found that it is valid for all but the most severe of bubble responses\[29, 68\]. Neglecting thermal and high Mach number effects, the gas within the nuclei bubble is assumed to behave in a polytropic manner where,

\[
p_{in}V_b^n = \text{Constant} \tag{3.4}
\]

where \( n \) is the polytropic exponent. The vapor within the bubble is assumed to exist at a constant partial pressure equal to the vapor pressure of the liquid phase. Since the bubble remains spherical at all times, the volume of the bubble is,

\[
V_b = \frac{4}{3}\pi r^3 \tag{3.5}
\]

where \( a \) is the bubble radius. If we neglect all mass transfer across the nuclei bubble interface, we can determine the internal pressure if the current radius and initial state of the bubble are known. The initial state of the bubble is given by the initial pressure outside of the bubble, \( p_{\infty,0} \), and the initial radius of the bubble, \( a_0 \). Assuming that we do know the initial state, the internal pressure is,

\[
p_{in} = p_{in,0} \left(\frac{a_0}{a}\right)^n \tag{3.6}
\]

The initial pressure is a function of the forces acting upon the nuclei bubble during
the initial state. The initial pressure can be written as a function of,

$$p_{\text{in},0} = f(p_{\infty,0} - p_v, a_0, S)$$ (3.7)

These variables which govern the size of a nuclei bubble are now added to the list of variables upon which $p_{\infty,i}$ depends. This gives,

$$p_{\infty,i} - p_v = f(U_J, D_J, \rho, \mu, a_i, S, p_{\infty,0} - p_v, a_0)$$ (3.8)

If you carefully examine a clear glass of water which has just been filled from the tap, you will notice that the water is hazy for a few seconds. This is because millions of tiny bubbles were formed when the glass was filled. Many of these bubbles will rise to the surface or dissolve into the water, but a very large number will persist indefinitely. A cavitation experiment conducted with this water will have this large number of nuclei bubbles available to enter into a region of low pressure and initiate cavitation. If the very same water is then processed in a way which removes a large portion of the nuclei bubbles, fewer nuclei bubbles will be available to cause cavitation. If the number is drastically reduced, the likelihood of observing cavitation may be significantly reduced. This might require the reference pressure to be reduced below the critical value until smaller bubbles, which will likely exist in higher numbers, begin to cavitate or until the liquid ruptures and forms significant pockets of vapor in the flow. It appears that the number density of nuclei bubbles, $N$, will be important to the magnitude of $p_{\infty,i}$. Adding $N$ to the functional relation for $p_{\infty,i}$ gives,

$$p_{\infty,i} - p_v = f(U_J, D_J, \rho, \mu, a_i, S, p_{\infty,0} - p_v, a_0, N)$$ (3.9)

Studies of rigid bubbles in vortical flows have shown that bubbles are drawn to the cores of vortices\cite{40, 41, 42, 46}. Johnson and Hsieh have shown that flows over headforms can result in a screening process where large bubbles are pushed away from the low pressure region on the surface of the headform\cite{36}. These studies suggest that the physical variables which govern the motion of nuclei bubbles may also be important to $p_{\infty,i}$. The spatial location of a nuclei bubble, $x_b$, is a function of the bubble velocity, $v$, which is in turn a function of the forces which
act upon the bubble. These forces, summarized in figure 3.3, include buoyancy, pressure gradients, and the difference in velocity between the bubble and the local flow. Aside from buoyancy, the other forces are functions of the same variables which are listed in equation 3.1. Buoyancy, however, is a function of gravitational acceleration, the difference in density between the fluid and the gas within the bubble, and the volume of the bubble (which is proportional to the radius). We can summarize the dependence of $\vec{x}_b$ as,

$$\vec{x}_b = f(U_J, D_J, \mu, \rho, \rho_b, g, a) \quad (3.10)$$

where $\rho_b$ is the density within the bubble and $g$ is the gravitational constant. Assuming that $p_{\infty,i}$ may be a function of the location of the nuclei bubble, equation 3.10 reveals that two additional dimensional variables, $\rho_b$ and $g$, will be important. Adding these to equation 3.9 gives,

$$p_{\infty,i} - p_v = f(U_J, D_J, \rho, \mu, a_i, S, p_{\infty,0} - p_v, a_0, N, \rho_b, g) \quad (3.11)$$

Figure 3.3: Sketch of the variables which are expected to influence the total force exerted on a nuclei bubble which affect the motion of the bubble.

Equation 3.11 provides the complete list of the physical variables which will be considered as having a significant influence on the value of $p_{\infty,i}$. These physical variables are listed in table 3.1 along with their dimensional units and corresponding basic dimensions. The basic dimensions chosen for this dimensional analysis exercise are mass ($\hat{M}$), length ($\hat{L}$), and time ($\hat{T}$). The Buckingham Pi Theorem states that the number of dimensionless groups which influence the physics is the difference between the number of physical variables that affect the problem and the number of basic dimensions needed to represent these physical variables.
Using this theorem, we expect that the problem is governed by no less than 9 dimensionless groups (12 physical variables minus 3 basic dimensions).

The process of finding the dimensionless groups involves choosing three repeating variables (the same number of repeating variables as basic dimensions). We will use $U_J$, $\rho$, and $\mu$ as the repeating variables for this exercise. The dimensionless $\pi_i$ groups, $\Pi_i$, are found by multiplying 1 non-repeating variable by the repeating variables to make the combination dimensionless. We will start with the first non-repeating variable, $p_{\infty,i} - p_v$.

$$\Pi_1 = p_{\infty,i} - p_v U_J^\alpha \rho^\beta \mu^\gamma \left( \frac{\hat{M}^1 \hat{L}^{-1} \hat{T}^{-2}}{(\hat{M}^0 \hat{L}^1 \hat{T}^{-1})^\alpha (\hat{M}^1 \hat{L}^{-3} \hat{T}^0)^\beta (\hat{M}^1 \hat{L}^{-1} \hat{T}^{-1})^\gamma} \right)$$

$$\alpha = -2$$
$$\beta = -1$$
$$\gamma = 0$$

$$\Pi_1 = \frac{p_{\infty,i} - p_v}{\rho U_J^2}$$

(3.12)

This first $\Pi$ group returns a dimensionless pressure. The process can be followed for the remaining non-repeating variables in table 3.1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
<th>Basic Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_{\infty,i} - p_v$</td>
<td>$N \ m^2$</td>
<td>$M^1 \hat{L}^{-1} \hat{T}^{-2}$</td>
</tr>
<tr>
<td>$U_J$</td>
<td>$m \ s^{-1}$</td>
<td>$M^0 \hat{L}^1 \hat{T}^{-1}$</td>
</tr>
<tr>
<td>$D_J$</td>
<td>$m$</td>
<td>$M^0 \hat{L}^1 \hat{T}^0$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>$kg \ m^{-3}$</td>
<td>$M^1 \hat{L}^{-3} \hat{T}^0$</td>
</tr>
<tr>
<td>$\mu$</td>
<td>$kg \ m^{-1} \ s^{-1}$</td>
<td>$M^1 \hat{L}^{-1} \hat{T}^{-1}$</td>
</tr>
<tr>
<td>$a_i$</td>
<td>$m$</td>
<td>$M^0 \hat{L}^1 \hat{T}^0$</td>
</tr>
<tr>
<td>$S$</td>
<td>$kg \ s^{-2}$</td>
<td>$M^1 \hat{L}^0 \hat{T}^{-2}$</td>
</tr>
<tr>
<td>$p_{\infty,0} - p_v$</td>
<td>$N \ m^2$</td>
<td>$M^1 \hat{L}^{-1} \hat{T}^{-2}$</td>
</tr>
<tr>
<td>$a_0$</td>
<td>$m$</td>
<td>$M^0 \hat{L}^1 \hat{T}^0$</td>
</tr>
<tr>
<td>$N$</td>
<td>$m^{-3}$</td>
<td>$M^{-3} \hat{L}^0 \hat{T}^0$</td>
</tr>
<tr>
<td>$\rho_b$</td>
<td>$kg \ m^{-3}$</td>
<td>$M^1 \hat{L}^{-3} \hat{T}^0$</td>
</tr>
<tr>
<td>$g$</td>
<td>$m \ s^{-2}$</td>
<td>$M^0 \hat{L}^1 \hat{T}^{-2}$</td>
</tr>
</tbody>
</table>

Table 3.1: Variables governing cavitation inception along with the corresponding dimensional units and basic dimensions
\[ \Pi_2 = D_J \ U^a \ \rho^\beta \ \mu^\gamma \ (=) \ \left( \hat{M}^0 \ \hat{L}^1 \ \hat{T}^0 \right) \ \left( \hat{M}^0 \ \hat{L}^1 \ \hat{T}^{-1} \right)^a \ \left( \hat{M}^1 \ \hat{L}^{-3} \ \hat{T}^0 \right)^b \ \left( \hat{M}^1 \ \hat{L}^{-1} \ \hat{T}^{-1} \right)^c \]

\[ \alpha = 1 \]
\[ \beta = 1 \]
\[ \gamma = -1 \]
\[ \Pi_2 = \frac{\rho D_J U_J}{\mu} = Re_J \]

\[ \Pi_3 = a_i \ U^a \ \rho^\beta \ \mu^\gamma \ (=) \ \left( \hat{M}^0 \ \hat{L}^1 \ \hat{T}^0 \right) \ \left( \hat{M}^0 \ \hat{L}^1 \ \hat{T}^{-1} \right)^a \ \left( \hat{M}^1 \ \hat{L}^{-3} \ \hat{T}^0 \right)^b \ \left( \hat{M}^1 \ \hat{L}^{-1} \ \hat{T}^{-1} \right)^c \]

\[ \alpha = 1 \]
\[ \beta = 1 \]
\[ \gamma = -1 \]
\[ \Pi_3 = \frac{\rho a_i U_J}{\mu} \]

\[ \Pi_4 = S \ U^a \ \rho^\beta \ \mu^\gamma \ (=) \ \left( \hat{M}^1 \ \hat{L}^0 \ \hat{T}^{-2} \right) \ \left( \hat{M}^0 \ \hat{L}^1 \ \hat{T}^{-1} \right)^a \ \left( \hat{M}^1 \ \hat{L}^{-3} \ \hat{T}^0 \right)^b \ \left( \hat{M}^1 \ \hat{L}^{-1} \ \hat{T}^{-1} \right)^c \]

\[ \alpha = -1 \]
\[ \beta = 0 \]
\[ \gamma = -1 \]
\[ \Pi_4 = \frac{S}{U_J \mu} \]

\[ \Pi_5 = p_{\infty,0} - p_e \ U^a \ \rho^\beta \ \mu^\gamma \ (=) \ \left( \hat{M}^1 \ \hat{L}^{-1} \ \hat{T}^{-2} \right) \ \left( \hat{M}^0 \ \hat{L}^1 \ \hat{T}^{-1} \right)^a \ \left( \hat{M}^1 \ \hat{L}^{-3} \ \hat{T}^0 \right)^b \ \left( \hat{M}^1 \ \hat{L}^{-1} \ \hat{T}^{-1} \right)^\gamma \]

\[ \alpha = -2 \]
\[ \beta = -1 \]
\[ \gamma = 0 \]
\[ \Pi_5 = \frac{p_{\infty,0} - p_e}{\rho \ U^2_J} \]

\[ (3.13) \]

\[ (3.14) \]

\[ (3.15) \]

\[ (3.16) \]
\[ \Pi_6 = a_0 U_j^a \rho^\beta \mu^\gamma = (\hat{M}^0 \hat{L}^1 \hat{T}^0) (\hat{M}^0 \hat{L}^1 \hat{T}^{-1})^a (\hat{M}^1 \hat{L}^{-3} \hat{T}^0)^b (\hat{M}^1 \hat{L}^{-1} \hat{T}^{-1})^c \]

\[ \alpha = 1 \]
\[ \beta = 1 \]
\[ \gamma = -1 \]
\[ \Pi_6 = \frac{\rho a_0 U_j}{\mu} \] (3.17)

\[ \Pi_7 = N U_j^a \rho^3 \mu^\gamma = (\hat{M}^0 \hat{L}^{-3} \hat{T}^0) (\hat{M}^0 \hat{L}^1 \hat{T}^{-1})^a (\hat{M}^1 \hat{L}^{-3} \hat{T}^0)^b (\hat{M}^1 \hat{L}^{-1} \hat{T}^{-1})^c \]

\[ \alpha = -3 \]
\[ \beta = -3 \]
\[ \gamma = 3 \]
\[ \Pi_7 = \frac{\mu^3 N}{\rho^3 U_j^a} \] (3.18)

\[ \Pi_8 = \rho_b U_j^a \rho^3 \mu^\gamma = (\hat{M}^1 \hat{L}^{-3} \hat{T}^0) (\hat{M}^0 \hat{L}^1 \hat{T}^{-1})^a (\hat{M}^1 \hat{L}^{-3} \hat{T}^0)^b (\hat{M}^1 \hat{L}^{-1} \hat{T}^{-1})^c \]

\[ \alpha = 0 \]
\[ \beta = -1 \]
\[ \gamma = 0 \]
\[ \Pi_8 = \frac{\rho_b}{\rho} \] (3.19)

\[ \Pi_9 = g U_j^a \rho^\beta \mu^\gamma = (\hat{M}^0 \hat{L}^1 \hat{T}^{-2}) (\hat{M}^0 \hat{L}^1 \hat{T}^{-1})^a (\hat{M}^1 \hat{L}^{-3} \hat{T}^0)^b (\hat{M}^1 \hat{L}^{-1} \hat{T}^{-1})^c \]

\[ \alpha = -3 \]
\[ \beta = -1 \]
\[ \gamma = 1 \]
\[ \Pi_9 = \frac{g \mu}{\rho U_j^a} \] (3.20)

These \( \Pi \) groups can be transformed into more recognizable quantities by multiplying by other \( \Pi \) groups. For example, \( \Pi_9 \) can be transformed into a reciprocal...
of a Froude number squared by multiplying through by $\Pi_2$.

$$\Pi'_9 = \Pi_9 \times \Pi_2 = \frac{g \mu}{\rho U^3_j} \times \frac{\rho U J D_J}{\mu} = \left( \frac{\sqrt{g D_J}}{U_J} \right)^2 = Fr^{-2} \quad (3.21)$$

Since the variable is dimensionless, the inverse of the square root is also dimensionless. Following this process, other recognizable dimensionless quantities can be derived, as shown in table 3.2. The final functional form for the problem of a cavitating jet is,

$$\frac{p_{\infty,i} - p_v}{\rho U^2_j} = f \left( Re_J, Fr, We, \frac{a_i}{D_J}, \frac{p_{\infty,0} - p_v}{\rho U^2_j}, \frac{a_0}{D_J}, N D^3_J, \frac{\rho_b}{\rho} \right) \quad (3.22)$$

The dimensionless dependent variable is a physical pressure difference which is normalized by the dynamic head of the jet flow. The value of the pressure difference is a function of only the jet Reynolds number, $\Pi_2$. The remaining terms arise from considerations of the bubble dynamics. This means that if cavitation inception occurs when the minimum pressure reaches the vapor pressure (i.e. bubbles are not important) then two flows with the same Reynolds numbers, but different length and velocity scales, would have constant values of $\frac{p_{\infty,i} - p_v}{\rho U^2_j}$. While it is ideal to design scale model experiments to operate at the same Reynolds number as the prototype flow, in reality this is not possible for many high Reynolds number problems. The scale model flow is limited by the extremely high velocities which are required to maintain $Re$. The velocity of a model is proportional to the ratio of the length scales of the prototype to the model. This can be shown by setting the model and prototype Reynolds numbers equal to one another and solving for the model velocity while holding the fluid properties constant. This gives,

$$U_m = U_p \left( \frac{D_p}{D_m} \right) \quad (3.23)$$

where the subscript “$m$” indicates a model value and a subscript “$p$” indicates a prototype value. High $Re$ problems will consist of high velocities and/or large dimensions. A scale model may have significantly smaller dimensions which require significantly larger velocities.

The large velocities required to maintain Reynolds number similarity in a scale
Table 3.2: Summary of dimensionless variables governing cavitation inception

<table>
<thead>
<tr>
<th>( \Pi )</th>
<th>( \frac{\rho \infty, i - \rho v}{\rho U_f^2} )</th>
<th>incipient cavitation number, ( \sigma, i )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Pi_2 )</td>
<td>( \frac{\rho U_f D_f}{\mu} )</td>
<td>jet Reynolds number, ( Re_f )</td>
</tr>
<tr>
<td>( \Pi_3 )</td>
<td>( \frac{a_i}{D_f} )</td>
<td>inception/jet size ratio</td>
</tr>
<tr>
<td>( \Pi_4 )</td>
<td>( \frac{\rho U_f^2 D_f}{S} )</td>
<td>jet Weber number, ( We )</td>
</tr>
<tr>
<td>( \Pi_5 )</td>
<td>( \frac{p_{\infty, 0} - \rho v}{\rho U_f^2} )</td>
<td>dimensionless bubble reference pressure</td>
</tr>
<tr>
<td>( \Pi_6 )</td>
<td>( \frac{a_0}{D_f} )</td>
<td>initial bubble/jet size ratio</td>
</tr>
<tr>
<td>( \Pi_7 )</td>
<td>( N D_f^3 )</td>
<td>nuclei density ratio</td>
</tr>
<tr>
<td>( \Pi_8 )</td>
<td>( \frac{\rho_b}{\rho} )</td>
<td>bubble/liquid density ratio</td>
</tr>
<tr>
<td>( \Pi_9 )</td>
<td>( \frac{U_f}{\sqrt{g D_f}} )</td>
<td>Froude number, ( Fr )</td>
</tr>
</tbody>
</table>

model can cause another problem. As the velocities increase, the value of \( p_{\infty, i} \) will also increase if \( \sigma_i \) is to remain constant. Scale model tests may require very high reference pressures to prevent cavitation from occurring. Because of the large velocities and reference pressures required to maintain Reynolds number similarity, many scale model experiments are performed at significantly lower \( Re \). However, this alters the dynamics of the liquid flow and may change the value of \( \sigma_i \).

The third \( \Pi \) group \( (a_i/D_f) \) is associated with the detection of cavitation inception. To maintain similarity of this parameter the definition of cavitation inception, \( a_i \), must change in proportion to the length scale of the flow. The next three \( \Pi \) groups determine the nuclei response relative to the pressure fluctuations within the jet flow. The fourth \( \Pi \) group \( (\rho U_f^2 D_f/S) \) is a Weber number defined by the jet flow conditions and the surface tension at the interface of a nuclei bubble. If Reynolds number similarity is held between a model and prototype scale, this quantity cannot remain constant unless the surface tension of the liquid is changed, which would require a change in temperature (a violation of our isothermal assumption) or a change in the fluids that comprise the bubble interface. The fifth and sixth \( \Pi \) groups \( ((p_{\infty, 0} - \rho v)/(\rho U_f^2), a_0/D_f) \) represent the variables governing the initial conditions for the nuclei bubble. We see that these physical variables must change with the length and velocity scales if \( \sigma_i \) is to remain a constant. For
practical reasons, it is common to use water for both scale model and prototype tests. If both tests are performed in fluid from the same source, we would expect the physical values of $a_0$ and $p_{\infty,0}$ to remain constant. Since the length scale and velocity scale change between the model and prototype, the fifth and sixth $\Pi$ groups cannot be constant. While this would probably be difficult to control, in theory it would be possible to obtain similarity in the sixth $\Pi$ group ($a_0/D_J$) by filtering out all of the larger nuclei bubbles from the scale model simulation. To be useful, this filtering would need to take place at a reference pressure defined by $\Pi_5$, ensuring that the remaining nuclei bubbles in the model flow have a similar initial state as the nuclei bubbles in the prototype flow.

The seventh $\Pi$ group ($N/D_J^3$) is related to the availability of a nuclei to cause a cavitation event. The number density of a given size of nuclei bubbles within a flow is expected to be on the order of $1 \times 10^3$ to $1 \times 10^4$ per cubic meter[69]. It is not entirely clear that this parameter is significant to most real-world applications, but may become important for very small scale models which represent very large flows. Regardless of its importance, if the same fluid is used in both the scale model and prototype flows, the number density of nuclei will not change, meaning that the seventh $\Pi$ group cannot remain constant.

The last two $\Pi$ groups are a consequence of the variables which affect the nuclei bubble motion within the fluid flow. The eighth $\Pi$ group ($\rho_b/\rho$) is the density ratio between the gas within the bubble and the surrounding liquid. Since most cavitation studies at both model and prototype scales are performed in water, the eighth $\Pi$ group will typically remain constant between model and prototype scales. The ninth $\Pi$ group ($Fr = U_J/\sqrt{gD_J}$) is proportional to the ratio between the jet velocity and the square root of the jet diameter. The $\Pi$ group is known as the Froude number. Jets which maintain Reynolds number similarity and use the same fluid will have an inverse relationship between the jet velocity and the diameter. Therefore, the Froude number cannot be held constant without changing the local gravitational acceleration (which is highly impractical). This means that Froude number and Reynolds number similarity cannot both be maintained.

The change in the dimensionless groups under certain limiting cases is shown in table 3.3. The dimensionless $\Pi$ groups will change when either Reynolds number or the Froude number is held constant and the same liquid is used for both
the model and the prototype. For each type of similarity, six of the dimensionless groups cannot be held constant. This indicates that $\sigma_i$, which is a function of all 8 $\Pi$ groups, will be influenced by changes to the length scale. Even though it has been the focus of more than 50 years worth of research, the specific relationship between $\sigma_i$ and these other groups is not understood and requires further study. The difficulty in controlling (or even measuring) many of these dimensionless groups provides motivation for developing a numerical tool to directly simulate the prototype flow instead of trying to experimentally determine the functional relationship for $\sigma_i$.

<table>
<thead>
<tr>
<th>Group</th>
<th>Constant $Re$</th>
<th>Constant $Fr$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Pi_2 = \frac{\rho U_j D_j}{\mu}$</td>
<td>constant</td>
<td>$O\left(\frac{D_m}{D_p}\right)^{3/2}$</td>
</tr>
<tr>
<td>$\Pi_3 = \frac{a_0}{D_j}$</td>
<td>$O\left(\frac{D_m}{D_p}\right)$</td>
<td>$O\left(\frac{D_m}{D_p}\right)$</td>
</tr>
<tr>
<td>$\Pi_4 = \frac{\rho U_j^2 D_j}{S}$</td>
<td>$O\left(\frac{D_m}{D_p}\right)$</td>
<td>$O\left(\frac{D_m}{D_p}\right)^2$</td>
</tr>
<tr>
<td>$\Pi_5 = \frac{\rho_{\infty} \theta - \rho_{\infty}}{\rho U_j^2}$</td>
<td>$O\left(\frac{D_m}{D_p}\right)^2$</td>
<td>$O\left(\frac{D_m}{D_p}\right)$</td>
</tr>
<tr>
<td>$\Pi_6 = \frac{a_0}{D_j}$</td>
<td>$O\left(\frac{D_m}{D_p}\right)$</td>
<td>$O\left(\frac{D_m}{D_p}\right)$</td>
</tr>
<tr>
<td>$\Pi_7 = N D_j^3$</td>
<td>$O\left(\frac{D_m}{D_p}\right)^3$</td>
<td>$O\left(\frac{D_m}{D_p}\right)^3$</td>
</tr>
<tr>
<td>$\Pi_8 = \frac{\rho_k}{\rho}$</td>
<td>constant</td>
<td>constant</td>
</tr>
<tr>
<td>$\Pi_9 = \frac{U_j}{\sqrt{g D_j}}$</td>
<td>$O\left(\frac{D_m}{D_p}\right)^{3/2}$</td>
<td>constant</td>
</tr>
</tbody>
</table>

Table 3.3: Expected change in the independent $\Pi$ groups as the fluid is held constant but the length scale is changed. The order of magnitudes represent the change of $\Pi_x$ at the prototype scale if $\Pi_x$ is known at the model scale. The subscripts “m” and “p” represent the values of the length scale for the model and prototype, respectively. The second column represents the expected change as the Reynolds number of the jet is held constant and the third column represents the expected change as the Froude number of the jet is held constant.
3.2 Additional Dimensionless Groups

Before moving on to other topics, I would like to note that the problem of cavitat
ion in a circular jet is even more complicated that what was discussed in sec-
tion 3.1. The dimensional analysis performed in the previous section assumed
that the liquid flow could be characterized by a single dimensionless Reynolds number.
Unfortunately, there is more than one length scale in the quiescent jet problem and
additional length and velocity scales associated with a co-flow jet. The additional
length scale for the quiescent jet is shear layer thickness, denoted by the variable \( \delta \). As discussed in chapter 2, the shear layer thickness is an important parameter
to the dynamics of the jet flow, but most of the previous studies of jet flows and
cavitation inception in jets have neglected to report this value. If this variable is
included in the dimensional analysis, an additional \( \Pi \) group is obtained which has the form,

\[
\Pi_{10} = \delta U_j^{\alpha} \rho^{\beta} \mu^{\gamma} (=) (\dot{M}^0 \dot{L}^1 \dot{T}^0) (\dot{M}^0 \dot{L}^{-1} \dot{T}^{-1})^{\alpha} (\dot{M}^1 \dot{L}^{-3} \dot{T}^0)^{\beta} (\dot{M}^1 \dot{L}^{-1} \dot{T}^{-1})^{\gamma}
\]

\[
\alpha = 1 \\
\beta = 1 \\
\gamma = -1 \\
\Pi_{10} = \frac{\rho \delta U_j}{\mu}, \text{ or } \frac{\delta}{D_j}
\]

From this result, we can conclude that \( \sigma_i \) is a function of at least nine other
dimensionless groups.

A co-flowing jet adds an additional velocity associated with the external flow, \( U_{co} \). There will also be an external shear layer thickness, \( \delta_{co} \). It is also plausible
that the distance between the shear layers (due to the thickness of the nozzle wall),
\( L_{co} \), will be important to the dynamics of the co-flow jet. The external velocity and
distance between the shear layers is illustrated in figure 3.4. These variables add
three more dimensionless groups to the problem which must also be held constant
to maintain dynamic similarity. The resulting \( \Pi \) groups are,

\[
\Pi_{11} = U_{co} U_j^{\alpha} \rho^{\beta} \mu^{\gamma} (=) (\dot{M}^0 \dot{L}^1 \dot{T}^0) (\dot{M}^0 \dot{L}^{-1} \dot{T}^{-1})^{\alpha} (\dot{M}^1 \dot{L}^{-3} \dot{T}^0)^{\beta} (\dot{M}^1 \dot{L}^{-1} \dot{T}^{-1})^{\gamma}
\]

\[
\alpha = -1 \\
\beta = 0 \\
\gamma = 0 \\
\Pi_{11} = \frac{U_{co}}{U_j}
\]
\[ \Pi_{12} = \delta_{co} U_j^\gamma \rho^\beta \mu^\alpha \left( \dot{M}_0^0 \dot{L}^1 \dot{T}^0 \right) \left( \dot{M}_0^1 \dot{L}^{-3} \dot{T}^0 \right)^\beta \left( \dot{M}_1^1 \dot{L}^{-1} \dot{T}^{-1} \right)^\gamma \]

\[ \alpha = 1 \]
\[ \beta = 1 \]
\[ \gamma = -1 \]
\[ \Pi_{12} = \frac{\rho \delta_{co} U_j}{\mu}, \text{ or } \frac{\delta_{co}}{D_j} \]

\[ \Pi_{13} = L_{co} U_j^\gamma \rho^\beta \mu^\alpha \left( \dot{M}_0^0 \dot{L}^1 \dot{T}^0 \right) \left( \dot{M}_0^1 \dot{L}^{-3} \dot{T}^0 \right)^\beta \left( \dot{M}_1^1 \dot{L}^{-1} \dot{T}^{-1} \right)^\gamma \]

\[ \alpha = 1 \]
\[ \beta = 1 \]
\[ \gamma = -1 \]
\[ \Pi_{13} = \frac{\rho L_{co} U_j}{\mu}, \text{ or } \frac{L_{co}}{D_j} \]

It is apparent that \( \sigma_i \) will be a function of twelve dimensionless \( \Pi \) groups for a cavitating co-flow jet. The full functional relationship for a cavitating jet is,

\[ \frac{p_{\infty,i} - p_v}{\rho U_j^2} = f \left( Re_J, Fr, We, \frac{a_i}{D_J}, \frac{p_{\infty,0} - p_v}{\rho U_j^2}, \frac{a_0}{D_J}, N, D_j^3, \frac{\rho_b}{\rho}, \frac{\delta}{D_J}, \frac{U_{co}}{U_j}, \frac{\delta_{co}}{D_j}, \frac{L_{co}}{D_j} \right) \]

(3.28)

It is not possible to conduct a scale model test where values of \( \Pi_2 \) through \( \Pi_{13} \) do not change from the prototype. The challenge for predicting the prototype incipient value of \( \Pi_1 \) from the scale model test data is to understand how \( \Pi_1 \) will change when each of the remaining \( \Pi \) groups are altered. Unfortunately, the set of \( \Pi \) groups is quite large and it has proved to be difficult to isolate the effects on \( \Pi_1 \) when the remaining \( \Pi \) groups change.

### 3.3 Categorization of Dimensionless Groups

Billet and Holl suggested that the dimensionless groups involved in cavitation inception be categorized by their specific effects on the problem\(^7\). Dimensionless groups which affect the bulk liquid flow were labeled under “type-1” parameters whereas the groups which affect the bubble growth process are labeled as “type-2” parameters. This concept can be used to reduce the number of dimensionless groups which must be considered when the physical variables are altered. For example, scale model flows which are dynamically similar to a prototype flow
should not be affected by the type-1 parameters. This has prompted researchers to look at type-2 parameters to explain scale effects in $\sigma_i$. Tests conducted using a single model at different Reynolds numbers may result in changes to $\sigma_i$ because of type-1 and type-2 parameters.

Billet and Holl considered only the physical variables which directly affect the fluid dynamics or bubble growth processes. The present dimensional analysis also considers the physical variables which affect the motion of nuclei bubbles, detection of nuclei bubbles, and the availability of nuclei bubbles. These additional physical variables, and the resultant dimensionless groups, can be categorized as “type-3” and “type-4” parameters. The type-3 parameters are defined to be those which influence the bubble motions relative to the fluid motions while type-4 parameters are defined as those which influence the ability to detect a cavitation event. The study of Johnson and Hsieh\[36\] showed that screening effects (type-3 effects) can be important to cavitation inception for headforms. $\sigma_i$ for headforms can then be functions of type-2, type-3, and type-4 parameters when dynamically-similar tests are conducted at different length scales, or type-1, type-2, and type-3 parameters when the same model is tested at different Reynolds numbers. The additional influence of type-3 parameters, which have not been considered before, may explain why it has been difficult to correlate $\sigma_i$ to either type-2 or type-3 parameters. Type-4 parameters should only contribute to $\sigma_i$ when the length scale of the flow is changed. The categorization of the thirteen dimensionless groups into these four
types is summarized in table 3.4.

<table>
<thead>
<tr>
<th>Type 1</th>
<th>Type 2</th>
<th>Type 3</th>
<th>Type 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re$</td>
<td>$We$</td>
<td>$\frac{\rho_b}{\rho}$</td>
<td>$\frac{\alpha_b}{D_j}$</td>
</tr>
<tr>
<td>$\frac{\delta}{D_j}$</td>
<td>$\frac{p_{\infty} - p}{\rho U_j^2}$</td>
<td>$Fr$</td>
<td>$N D_j^3$</td>
</tr>
<tr>
<td>$\frac{U_{\infty}}{U_j}$</td>
<td>$\frac{\alpha_0}{D_j}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\frac{\delta_{\infty}}{D_j}$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\frac{L_{\infty}}{D_j}$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3.4: Table of dimensionless groups and the category to which they are most closely associated. Type-1 parameters influence the dynamics of the bulk flow, Type-2 parameters influence the size of the nuclei bubbles, Type-3 parameters influence the motion of the nuclei bubbles, and Type-4 parameters influence the likelihood of detecting incipient cavitation.

3.4 Summary

The dimensional analysis performed in this chapter indicates that the incipient cavitation number for a high-Reynolds number circular jet is a function of up to 12 other dimensionless parameters. All of these parameters cannot be held constant between a model and a prototype flow. Because of this, the cavitation inception number is likely to change as the physical properties of the flow are altered. An experimental study to determine the relationship between all of these parameters is a daunting task that would be difficult and expensive to accomplish. Since the functional relationship between $\sigma_i$ and the remaining dimensionless groups is not known, and will not be easy to obtain, another means of predicting $\sigma_i$ for prototype flows is needed. The remaining chapters will address the mathematical and numerical methods needed to obtain numerical simulations of cavitation inception. These numerical simulations will attempt to directly solve for $\sigma_i$ without knowing the functional relationships between $\sigma_i$ and the remaining dimensionless groups. If successful, these simulations can also be used to examine the functional relationships for $\sigma_i$ and provide insight into the experimentally observed scale effects.
Mathematical Models

The goal of this thesis is to understand the origins of cavitation scale effects so that they may be predicted. To achieve this goal, mathematical models of the underlying physics must be developed. Once developed, these mathematical equations may be analyzed to determine possible sources for cavitation scale effects and can be numerically solved on a computer. The success of the cavitation simulations relies heavily on the ability of the mathematical models to predict the behavior of incipient cavitation. This chapter derives the mathematical models which will be used to simulate cavitation inception. The equations which govern the motion of the liquid will be discussed first. Following the discussion of the bulk liquid models, the mathematical models for cavitation nuclei bubbles will be derived. The derivation of these equations from first principles will provide insight into the underlying assumptions of the equations and the limitations of their use. These mathematical models will be examined to understand the physical parameters that most significantly affect their solution. This chapter will also examine how these physical parameters change between model and prototype flows where similarity in the bulk flow is maintained. Different mathematical definitions of cavitation inception will also be discussed.

4.1 Fluid Dynamics: Type-1 Scale Effects

A full derivation of the equations which govern the bulk fluid flow will not be presented here. Derivations are discussed in great detail in most any fluid mechanics
text book such as the ones by Panton [70], Batchelor [31], Kundu and Cohen [71], or Cengel and Cimbala [72]. The focus of this section will instead be on the assumptions and limitations of the governing equations as presented in the sources mentioned.

Let’s begin by limiting our discussion of cavitation inception to Newtonian incompressible liquids such as water. Let’s further assume that the temperature in the flow is spatially and temporally uniform. The governing equations for fluids such as this are the Navier-Stokes (N-S) equations, which include the continuity equation,

\[ \frac{\partial u_i}{\partial x_i} = 0, \]  

(4.1)

and the momentum equations,

\[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j}. \]  

(4.2)

These equations are presented in index notation form where the subscripts “i” and “j” indicate the coordinate direction of the vector quantity with which they are associated. The values of i and j can be 1, 2, or 3, which represent the Cartesian coordinates x, y, and z, respectively. Repeated indices within a term imply summation over all coordinate directions for that index. The vector quantities in equations 4.1 and 4.2 are x, the spatial location, and u, the liquid velocity. The variable t represents the time associated with the values of all other quantities, p is the local pressure of the liquid, \( \rho \) is the local density of the liquid, and \( \nu \) is the local kinematic viscosity of the liquid. The assumption of a spatially and temporally isothermal flow allow both density (\( \rho \)) and viscosity (\( \nu \)) to be treated as constants and eliminates any thermodynamic components of pressure.

It is often convenient to examine the governing equations in their dimensionless form. The dimensionless form has the advantage that there are a reduced number of variables and these variables represent the “universal” or “natural” variables that govern the solution to the equations [70]. Thus, human-invented scales are removed from the physics. For this reason, only the nondimensional form of these governing equations will be considered in the remainder of this work. The nondimensional form of the governing equations can be found by defining a dimensionless length
and time scale as,

$$\tilde{x}_i = \frac{x_i}{D_J},$$

(4.3)

and

$$\tilde{t} = \frac{t}{(D_J/U_J)},$$

(4.4)

Using these dimensionless length and time scales, a dimensionless velocity scale can be defined as,

$$\tilde{u}_i = \frac{u_i}{U_J}.$$ 

(4.5)

The characteristic length and velocity scales in the definitions of the dimensionless variables are $U_J$, the mean velocity of the liquid exiting the jet, and $D_J$, the diameter of the jet. Multiplying the continuity equation (equation 4.1) by $D_J/U_J$ and substituting the resulting dimensionless quantities results in the dimensionless continuity equation,

$$\frac{\partial \tilde{u}_i}{\partial \tilde{x}_i} = 0.$$ 

(4.6)

The momentum equation can be nondimensionalized in a similar fashion by multiplying equation 4.2 with $D_J/U_J^2$. This gives the dimensionless continuity equation,

$$\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial \tilde{x}_j} = - \frac{\partial \tilde{p}}{\partial \tilde{x}_i} + \frac{1}{Re_f} \frac{\partial^2 \tilde{u}_i}{\partial \tilde{x}_j \partial \tilde{x}_j}.$$ 

(4.7)

where $\tilde{p}$ is a dimensionless pressure defined as,

$$\tilde{p} = \frac{p}{\rho U_J^2}.$$ 

(4.8)

and $Re_f$ is the Reynolds number of the jet flow defined as,

$$Re_f = \frac{D_J U_J}{\nu}.$$ 

(4.9)

Equations 4.6 and 4.7 are the dimensionless form of the Newtonian, incompressible N-S equations.

The dimensionless N-S equations represent the flow in a small, finite region of continuous fluid. To solve these equations on a computer, the flow domain must be discretized into small regions which are each governed by their own set
of N-S equations. Unfortunately, computing resources are limited and the length of the discretized regions are typically much larger than the smallest turbulent eddies of the flow. Solving the N-S equation on such a coarse grid has the effect of filtering out all of the turbulent fluctuations which are too small to be resolved. The consequences for not resolving the small turbulent fluctuations can be evaluated by decomposing the velocity into a filtered velocity component, $\tilde{u}$, and an unresolved residual velocity, $u'$. The substitution is,

$$\tilde{u} = \bar{u} + u'.\quad (4.10)$$

Equation (4.10) is substituted into the continuity equation (eq. 4.6). The results are then filtered to obtain an equation for the velocity field which can be resolved. The filter operation results in,

$$\frac{\partial \tilde{u}_i}{\partial t} = 0.\quad (4.11)$$

The results of filtering show that the continuity equation holds for the resolved portion of the velocity. Equation (4.10) is must also be substituted into the momentum equation (eq. 4.7). Filtering the resulting momentum equation gives,

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{1}{Re_f} \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} \left( \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j - \frac{2}{3} \bar{k} \delta_{ij} \right),\quad (4.12)$$

where $\bar{p}$ is the filtered pressure field, $\bar{k}$ is the dimensionless turbulent kinetic energy, and $\delta_{ij}$ is the Kronecker delta function. Filtering the momentum equation has introduced several new terms. While the quantity $\bar{u}_i \bar{u}_j$ can be resolved, the quantity $\bar{u}_i \bar{u}_j$ represents the filtered product of the full velocity field. Since the full velocity field is unknown, the product of the full velocity field is also unknown. The result of the filter operation shows that the resolved portion of the velocity is dependent upon the unresolved residual velocity. This is the classic closure problem where there is too many variables and not enough equations.

In the present work, the closure problem is dealt with by approximating the anisotropic residual stress terms with a linear closure model. The linear closure
model substitution is,

\[-\left( \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j - \frac{2}{3} \bar{k} \delta_{ij} \right) = \frac{1}{Re_t} \left( \frac{\partial \bar{u}_i}{\partial \bar{x}_j} + \frac{\partial \bar{u}_j}{\partial \bar{x}_i} \right) - \frac{2}{3} \bar{k} \delta_{ij}. \quad (4.13)\]

where $Re_t$ is the turbulent Reynolds number. Substituting this into equation 4.12 gives,

\[\frac{\partial \bar{u}_i}{\partial \bar{t}} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial \bar{x}_j} = -\frac{\partial \bar{P}}{\partial \bar{x}_i} + \left( \frac{1}{Re_f} + \frac{1}{Re_t} \right) \frac{\partial^2 \bar{u}_i}{\partial \bar{x}_j \partial \bar{x}_j} + \left( \frac{\partial}{\partial \bar{x}_j} \frac{1}{Re_t} \right) \left[ \frac{\partial \bar{u}_i}{\partial \bar{x}_j} + \frac{\partial \bar{u}_j}{\partial \bar{x}_i} \right], \quad (4.14)\]

where $\bar{P}$ is,

\[\bar{P} = \bar{p} + \frac{2}{3} \bar{k}. \quad (4.15)\]

Equations 4.11 and 4.14 represent the set of equations which govern the resolved (filtered) velocity and pressure field. This set of equations is still not closed since a new variable, $Re_t$, was introduced by the linear closure model. To close the system of equations, the turbulent Reynolds number is modeled using the blended $k - \epsilon/k - \omega$ two-equation turbulence model provided by Menter [62]. In this model the turbulent Reynolds number is given by,

\[Re_t = \frac{\tilde{\omega}_d}{\bar{k}} \quad (4.16)\]

where $\tilde{\omega}_d$ is the dimensionless specific dissipation rate. The values of $\bar{k}$ and $\tilde{\omega}_d$ are obtained from the solution to the partial differential equations,

\[\frac{\partial \bar{k}}{\partial \bar{t}} + \left[ \bar{u}_j - \sigma_k \frac{\partial}{\partial \bar{x}_j} \left( \frac{1}{Re_t} \right) \right] \frac{\partial \bar{k}}{\partial \bar{x}_j} - \frac{1}{Re_t} \frac{\partial^2 \bar{k}}{\partial \bar{x}_j \partial \bar{x}_j} + s_k = 0 \quad (4.17)\]

for turbulent kinetic energy and

\[\frac{\partial \tilde{\omega}_d}{\partial \bar{t}} + \left[ \bar{u}_j - \sigma_\omega \frac{\partial}{\partial \bar{x}_j} \left( \frac{1}{Re_t} \right) \right] \frac{\partial \tilde{\omega}_d}{\partial \bar{x}_j} - \frac{1}{Re_t} \frac{\partial^2 \tilde{\omega}_d}{\partial \bar{x}_j \partial \bar{x}_j} + s_\omega = 0 \quad (4.18)\]
for specific dissipation. The variables $s_k$ and $s_\omega$ are defined as,

$$s_k = R_k \left( \beta^* \bar{\omega}_d \bar{k} - G \right)$$

(4.19)

and

$$s_\omega = R_\omega \left[ \beta \bar{\omega}_d^2 + 2(1 - F_1) \frac{\sigma_\omega}{\bar{\omega}_d} \frac{\partial \bar{k}}{\partial \bar{x}_j} \frac{\partial \bar{\omega}_d}{\partial \bar{x}_j} - \gamma \frac{\bar{\omega}_d}{k} \right].$$

(4.20)

In these equations, $R_k$ and $R_\omega$ are,

$$R_k = \frac{1}{Re_f} + \frac{\sigma_k}{Re_l}$$

(4.21)

and

$$R_\omega = \frac{1}{Re_f} + \frac{\sigma_\omega}{Re_l}$$

(4.22)

The variable $G$ is defined as,

$$G = \frac{1}{Re_l} \left[ \left( \frac{\partial \bar{u}_1 \partial \bar{u}_2}{\partial \bar{x}_1 \partial \bar{x}_2} \right)^2 + \left( \frac{\partial \bar{u}_1 \partial \bar{u}_3}{\partial \bar{x}_1 \partial \bar{x}_3} \right)^2 + \left( \frac{\partial \bar{u}_2 \partial \bar{u}_3}{\partial \bar{x}_2 \partial \bar{x}_3} \right)^2 \right]$$

$$+ 2 \left( \frac{\partial \bar{u}_1}{\partial \bar{x}_1} \right)^2 + 2 \left( \frac{\partial \bar{u}_2}{\partial \bar{x}_2} \right)^2 + 2 \left( \frac{\partial \bar{u}_3}{\partial \bar{x}_3} \right)^2$$

(4.23)

and $F_1$ is a blending function which is designed to be 1 in the sublayer and logarithmic regions of the boundary layers and 0 in the wake regions. The formula for $F_1$ is,

$$F_1 = \tanh \left( \left\{ \text{MIN} \left[ \text{MAX} \left( \frac{\sqrt{k}}{0.09 \bar{\omega}_d \delta^{1/3}} \frac{500}{Re_f \bar{\omega}_d \delta^2} \frac{4\sigma_\omega \bar{k}}{CD_{k\omega} \delta^2} \right), 10^{-20} \right] \right\} \right)$$

(4.24)

where $CD_{k\omega}$ is,

$$CD_{k\omega} = \text{MAX} \left( 2\sigma_\omega \frac{1}{\bar{\omega}_d} \frac{\partial \bar{k}}{\partial \bar{x}_j} \frac{\partial \bar{\omega}_d}{\partial \bar{x}_j} ; 10^{-20} \right).$$

(4.25)

The remaining model constants are listed in table 4.1. The value to be used in equations 4.17 and 4.18 is determined as a weighted average based upon the value
of the blending function. The specific form of the weighted average is,

\[ \phi = F_1 \phi_1 + (1 - F_1) \phi_2 \]  \hspace{1cm} (4.26)

where \( \phi_1 \) and \( \phi_2 \) are the corresponding values which are listed in table 4.1

<table>
<thead>
<tr>
<th>( \phi )</th>
<th>( \phi_1 )</th>
<th>( \phi_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_k )</td>
<td>0.5</td>
<td>1.0</td>
</tr>
<tr>
<td>( \sigma_\omega )</td>
<td>0.5</td>
<td>0.856</td>
</tr>
<tr>
<td>( \beta )</td>
<td>0.075</td>
<td>0.0828</td>
</tr>
<tr>
<td>( \beta^* )</td>
<td>0.09</td>
<td>0.09</td>
</tr>
<tr>
<td>( \gamma )</td>
<td>0.553</td>
<td>0.4403</td>
</tr>
</tbody>
</table>

Table 4.1: Blended \( k - \epsilon/k - \omega \) model constants.

At this point we have the mathematical equations which will be used to represent the bulk liquid flow of the jet. However, the purpose of this thesis is to examine cavitation inception. Thus we must consider the existence of cavitation, or voids, within the liquid. If the volume of the cavitation is small with respect to the resolved flow field, then the existence of the cavitation will have a negligible impact upon the flow. If, however, the volume of the cavitation becomes significant with respect to the resolved flow, we would expect that the existence of cavitation will change the flow solution. When this occurs, the cavitation and the flow are two-way coupled and the problem becomes much more difficult to solve. Recall however, that the focus of this work is on cavitation inception, implying that we are concerned with physics which are typically undetectable. Only at the point of inception is a vapor cavity (just barely) detectable. This implies that up until the point of inception, the cavitation nuclei are very small and will have negligible influence on the bulk liquid dynamics.

For the remainder of this work we will proceed under the assumption that the fluid dynamics and cavitation are one-way coupled. This provides for a significant simplification in the mathematical treatment of cavitation inception, but at the cost of generality. As these mathematical models are solved, we must keep in mind that we are implicitly assuming that the cavitation nuclei do not have a
significant impact on the bulk liquid flow. In the case of cavitation inception in small model-scale flows, this assumption may not hold true.

4.2 Radial Bubble Dynamics: Type-2 Scale Effects

Research into cavitation inception indicates that for most real flows (i.e. experiments where the water supply has not undergone extreme measure to remove all impurities) cavitation inception is caused by the growth of preexisting nuclei bubbles which are found in the liquid. The dynamics of these nuclei bubble are two-fold: changes in nuclei bubble volume and changes in the bubble trajectory. While these dynamics are interrelated, meaning that changes in bubble volume will change how the bubble moves, we will consider these dynamic changes independently, beginning with the change in volume.

In deriving the mathematical equations which predict the volume of a nuclei bubble, it is helpful to make some limiting assumptions to simplify the mathematics. It is important though that these simplifying assumptions are representative of the actual physics to ensure that the predictions are as accurate as possible. One of the most helpful simplifying assumption that can be made is to assume that the nuclei bubbles remain spherical at all times. This assumption significantly reduces the degrees of freedom for the problem, constraining it to one spatial dimension, $r$. Of course we need to make sure that this assumption is valid by comparing with experimental data. Fortunately, experimental data indicates that small nuclei bubbles in water are indeed nearly spherical except during extreme changes in volume (violent growth or collapse) or in the presence of extreme velocity gradients (relative to the nuclei bubble size) in the surrounding fluid. When a bubble is exposed to these extreme events, the interface of the bubble becomes circumferentially unstable which leads to a breakup of the bubble. Aside from numerical solutions, it is not possible to analytically calculate anything but very small perturbations about a spherical bubble. In any case, a violent collapse and expansion of a nuclei bubble is typically associated with a cavitation event. Since this is a limiting case for our flow, we should incur only minor errors by neglecting circumferential per-
turbations from our mathematical models. Furthermore, the assumption that the fluid flow and cavitation inception are one-way coupled limits us to considering flows where the nuclei bubbles remain small in comparison to the resolved flow structures. Therefore we must violate the one-way coupled assumption (or have very weak surface tension) to encounter a situation where small nuclei bubbles are exposed to strong velocity gradients.

As a complimentary assumption, we can assume that the velocity of the liquid surrounding the nuclei bubble is constrained to the radial direction (i.e. $u_\theta = u_\phi = 0$). This assumption is appropriate for nuclei bubbles in a stationary fluid and nuclei bubbles in a traveling liquid where the nuclei bubble velocity is the same as the local fluid velocity and there are no velocity gradients across the bubble surface. Unfortunately, nuclei bubbles often have velocities which are different than the local fluid velocity. The difference between the local fluid velocity and the bubble velocity is called the “slip velocity”, but it is not to be confused with a non-zero velocity at the bubble surface. The slip velocity will cause the pressure forces on the bubble to be nonuniform around the surface which can lead to a non-spherical bubble shape. However, in the limit of creeping flow the average pressure force on the nuclei bubble, calculated by integrating the pressure over the surface of the nuclei bubble and dividing by the surface area, is the same as the free-stream pressure. So while the external pressure is expected to be nonuniform, it is not expected to have a significant effect on the equivalent size of the nuclei bubble. As the velocity increases, the average pressure on the nuclei bubble will no longer be the same as the local free-stream pressure. However, the Reynolds number of the flow around the nuclei bubble is expected to remain small (i.e. $Re_b < 100$), indicating that the errors incurred by neglecting the non-radial velocity components will be small.

Another assumption which will help us in developing the governing equations for the nuclei bubbles is the assumption that the liquid outside of the bubble behaves as an incompressible Newtonian fluid. This assumption is consistent with those of the bulk flow equations and allows for compressibility effects of the outer fluid to be neglected. With this assumption, the liquid flow outside of the bubble is governed by the same N-S equations as the bulk liquid flow (although in spherical coordinates). A situation of concern is the violent collapse of the spherical nuclei
bubbles. Under the conditions of a violent collapse, the energy of the collapsing bubble is focused on an ever smaller region of space. As such, the temperatures, pressures, and velocities of the gas and liquid which create the bubble will rapidly increase. It is quite possible for the velocity of the bubble interface to exceed the speed of sound for the liquid, making the incompressibility assumption incorrect. The solution for a collapsing bubble given by the incompressible model equations have been compared with the solution from more advanced model equations which take into account temperature and compressibility effects[29]. Qualitatively, the incompressible models give very similar results to the advanced models, although damping of the nuclei motions is significantly under-predicted as one might expect since the model neglects the compressibility and thermal effects. Another problem with the violent collapse is that the bubble interface becomes unstable and leads to a breakup of the nuclei bubble. It is the spherical symmetry that focuses the energy of a collapsing nuclei to ever smaller volumes. As nuclei bubbles become unstable and break up, the energy is dispersed causing temperatures, pressures, and velocities to be limited. Since it is not clear that considering compressibility effects will lead to a more accurate model, they will be neglected.

The previous assumptions have addressed the conditions in the liquid phase outside of the nuclei bubble. The behavior of the interior contents of the bubble must also be addressed. Assume that the nuclei bubble consists of a mixture of gas and vapor. The gas content will typically consist of the gases dissolved in the surrounding liquid as well as any gases present exterior to the liquid source. For this study it will be assumed that the gas within the nuclei bubbles is air. This seems to be a reasonable assumption since air will be present at the free surface of most water sources (oceans, laboratories, etc.) and will likely be a significant portion of the dissolved gas within the water. It will also be assumed that the partial pressure due to vapor within the bubble remains constant as the bubble changes volume. The partial pressure of the gas within the bubble will be assumed to follow a polytropic process. Before we can consider an equation which governs the internal pressure, we need to know if gas is released from or absorbed into solution with the liquid surrounding the bubble. The physics of this depend upon the concentration of dissolved gas within the liquid surrounding the nucleus and the pressure of the gas within the bubble, among other things. However,
nuclei bubbles have been found to persist within a fluid indefinitely, even though
the gas within the nuclei bubble has had ample time and favorable conditions to
dissolve into the fluid. Evidence indicates that the interface of real nuclei bubbles
attracts surfactants which tend to create an impermeable membrane surface which
prevents mass transfer from occurring. It is not clear how these impermeable
membranes react as the nuclei bubble changes volume, but it is not unreasonable
to assume that as the bubble grows in size, the surfactants are spread over a much
larger surface and no longer constitute an impermeable interface. Even if mass
transfer occurs with the growth of a nuclei bubble, the equations governing mass
transfer through these membranes are unknown, along with the typical makeup
of the surfactants. Even if the surfactants are ignored and rectified diffusion is
considered, calculations estimate that a bubble with an initial radius of 1\,mm in
water at 20°\,C at 1 atmosphere of pressure will require about \(1 \times 10^6\) seconds to
double in radius. A nuclei bubble with an initial radius of 10\,µm under the same
conditions would require about \(1 \times 10^2\) seconds to double in size. For most flows,
the time span with which we are concerned with the bubble is on the order of a
few seconds at most. Thus the bubbles will not have enough time to experience
significant growth due to mass transfer. Due to the slow process of diffusion and
evidence for the existence of surfactant membranes, mass transfer effects will be
neglected. Therefore, the amount of gas within the nuclei bubble is constant and
the partial pressure is governed by the polytropic relation,

\[ p_g V^n = \text{Constant} \quad (4.27) \]

where \(p_g\) is the partial pressure of the gas within the nuclei bubble, \(V\) is the volume
of the nuclei bubble, and \(n\) is the polytropic exponent which is 1 for isothermal
compression and expansion and equal to the gas constant for adiabatic compression
and expansion. Although implicit in the polytropic assumption, we can explicitly
state that the mixture of gas and vapor within the nuclei bubble is assumed to
remain homogeneous and isotropic at all times.

Using all of these assumptions, the governing equations for cavitation inception
can be developed. To begin, assume spherical symmetry and a bubble centered at
the origin of a spherical coordinate system with coordinate axes \(r, \theta,\) and \(\phi\). To
avoid confusion, the symbol “a” will be used to represent the bubble radius whereas the symbol r will represent the radial coordinate direction. The fluid surrounding the nuclei bubble is described by the N-S equations in spherical coordinates. Conservation of mass in spherical coordinates is,

$$\frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 u_r \right) + \frac{1}{r \sin \theta} \frac{\partial}{\partial \theta} (\sin \theta \ u_\theta) + \frac{1}{r \sin \theta} \frac{\partial}{\partial \phi} (u_\phi) = 0. \quad (4.28)$$

The second and third term of equation 4.28 are zero because of radial symmetry which leaves only terms with radial velocity components. The results of equation 4.28 indicate that the derivative of $r^2 u_r$ with respect to r must be zero. This is only true if,

$$r^2 u_r = C_1 \quad (4.29)$$

where $C_1$ is a constant that can be evaluated at the bubble surface, $r = a$, where a is a function of time ($a = a(t)$). Using this boundary condition gives,

$$C_1 = a^2 u_a = a^2 \frac{da}{dt}. \quad (4.30)$$

where $u_a$ is the radial velocity of the bubble surface. Substituting equation 4.30 into equation 4.29 and solving for $u_r$ gives,

$$u_r = \frac{a^2}{r^2} \frac{da}{dt}. \quad (4.31)$$

The result is an equation which provides the radial velocity at any location, r, outside of the bubble if the size (a) and velocity (da/dt) of the bubble interface are known.

Now consider conservation of momentum for the liquid outside of the bubble. The momentum equation in spherical coordinates is,
\[
\frac{1}{r^2 \sin^2(\theta)} \frac{\partial^2 u_r}{\partial \phi^2} - 0 - \frac{2}{r^2 \sin(\theta)} \frac{\partial}{\partial \theta} \left( u_\theta \sin(\theta) \right) - \frac{2}{r^2 \sin(\theta)} \frac{\partial u_\phi}{\partial \phi} = 0 \quad (4.32)
\]

where all velocity components other than \( u_r \) are assumed to be zero. Gathering all of the nonzero terms and placing all terms with velocity on the right-hand side gives,

\[
-\frac{1}{\rho} \frac{\partial p}{\partial r} = \frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} - \nu \left[ \frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 \frac{\partial u_r}{\partial r} \right) - \frac{2 u_r}{r^2} \right]. \quad (4.33)
\]

These results provide the pressure gradient at any radial location, \( r \). Substituting the continuity equation (equation 4.31) into the momentum equation (equation 4.33) and taking the derivatives where possible gives,

\[
-\frac{1}{\rho} \frac{\partial p}{\partial r} = \frac{2 a}{r^2} (\dot{a})^2 + \frac{a^2}{r^2} \ddot{a} - \frac{2 a^4}{r^5} (\dot{a})^2. \quad (4.34)
\]

where a dot over the radius indicate a derivative with respect to time (i.e. \( \dot{a} = da/dt \)) and two dots indicate a second derivative with respect to time. Notice that enforcing continuity causes the viscous terms in equation 4.33 to cancel out. Equation 4.34 can be integrated from the bubble surface (\( r = a \)) to the far field (\( r = \infty \)) to solve for the pressure just outside the bubble surface. The integration is,

\[
-\frac{1}{\rho} \int_{p_a}^{p} dp = \int_{a}^{\infty} \left[ \frac{2 a}{r^2} (\dot{a})^2 + \frac{a^2}{r^2} \ddot{a} - \frac{2 a^4}{r^5} (\dot{a})^2 \right] dr \quad (4.35)
\]

which gives,

\[
\frac{p_a - p}{\rho} = a \ddot{a} + \frac{3}{2} (\dot{a})^2 \quad (4.36)
\]

Equation 4.36 is an equation which contains the unknowns \( p_a \), along with the bubble radius and its first and second derivatives. A force balance on the dynamic surface of the nuclei bubble will allow us to make a substitution for \( p_a \), reducing the number of unknowns by one.

Consider the forces acting on the surface of an expanding bubble, illustrated in figure 4.1.
Mathematically, the radial force per unit area \( F_r \) on the surface of the bubble is,
\[
\sum F_r = (\sigma_{rr})_{r=a} + p_b - \frac{2S}{a},
\]
(4.37)
where \( p_b \) is the pressure inside of the bubble, \( 2S/a \) is the force due to surface tension within a spherical surface, \( (\sigma_{rr})_{r=a} \) is the force due to stresses within the fluid acting on the bubble surface, and \( S \) is the surface tension due to the interface between the fluid outside the bubble and the gas within the bubble. The surface stresses in spherical coordinates reduce to,
\[
(\sigma_{rr})_{r=a} = -p_a + 2\mu \frac{du_r}{dr}
\]
(4.38)
assuming spherical symmetry of the flow. Equation 4.38 shows that the external force on the bubble is not simply due to the pressure outside of the bubble. There is also a component of viscous stress which arises from the radial expansion or contraction of the bubble. Substituting equation 4.31 for \( u_r \) and equation 4.38 into equation 4.37 gives the sum of the forces acting on the bubble surface which is
\[
\sum F_r = -p_a - \frac{4\mu \dot{a}}{a} + p_b - \frac{2S}{a},
\]
(4.39)
Since the bubble interface has no mass, the sum of the forces across this interface must be equal to zero. This provides a means of solving for the pressure outside of the bubble surface, \( p_a \), which can then be substituted into equation 4.36. This gives the result,
\[
\frac{p_b - p}{\rho} = a\ddot{a} + \frac{3}{2} (\dot{a})^2 + 4\nu \dot{a} \frac{\dot{a}}{a} + \frac{2S}{\rho a},
\]
(4.40)
which is the governing equation for a spherical nuclei bubble driven by the pressure difference between the local fluid pressure, $p$, and the pressure inside of the bubble, $p_b$.

While the unknown pressure on the outer surface of the bubble interface has been eliminated, it has been replaced with the unknown pressure inside of the bubble. Assuming that the bubble is filled with a mixture of gas and vapor, the pressure inside the bubble is the sum of the partial pressures. It was stated earlier that the partial pressure of the vapor, $p_v$, is assumed to be constant and equal to the vapor pressure of the liquid outside the bubble. The gas phase is assumed to behave in a polytropic manner under compression and expansion. The relation between the gas pressure, $p_g$ and the volume of the bubble is listed by equation 4.27. Since the nuclei bubbles are assumed to remain spherical, the volume of a bubble is given by,

$$ V = \frac{4}{3} \pi a^3 \quad (4.41) $$

which is the equation for the volume of a sphere with radius $a$. Using the polytropic relation (equation 4.27) and the volume of the bubble (equation 4.41), the pressure of the gas within the bubble is,

$$ p_b = p_0 \left( \frac{a_0}{a} \right)^{3n} + p_v \quad (4.42) $$

where $p_0$ is the initial pressure within the nuclei bubble and $a_0$ is the initial radius of the nuclei bubble. This result can be substituted into the governing equation for the dynamics of a spherical nuclei bubble, equation 4.40, to get the equation which is known in the bubble dynamics literature as the Rayleigh-Plesset (R-P) equation. The equation is a second-order, inhomogeneous, ordinary differential equation (ODE) which has the form,

$$ \frac{p_v - p}{\rho} = a\ddot{a} + \frac{3}{2} (\dot{a})^2 + 4\nu \frac{\dot{a}}{a} + \frac{2S}{\rho a} - \frac{p_0}{\rho} \left( \frac{a_0}{a} \right)^{3n} \quad (4.43) $$

The first two terms come from the Navier-Stokes equations for the liquid flow outside of the nuclei bubble and are associated with the inertia of the liquid surrounding the bubble. The third term on the left-hand side came from the boundary conditions on $p_a$ and represents the viscous damping which occurs due to the ex-
pansion or contraction of the bubble surface. The fourth term on the left-hand side represents the force associated with surface tension which is inversely proportional to the nuclei bubble radius. The last term on the left-hand side represents the pressure of the gas within the nuclei bubble and is inversely proportional to the radius to the $3n$ power. The gas pressure changes rapidly as the nuclei bubble expands and contracts, acting as a spring which can cause the nuclei bubble to oscillate in time. Finally, the right-hand side contains the term which is responsible for driving the change in the bubble radius. The local pressure in the fluid surrounding the bubble, $p$, is changing in time which causes the radius to respond according to the terms on the left-hand side of the equation. Depending on the rate with which the bubble radius responds, oscillations in the bubble radius may begin which will continue until viscosity damps them out.

### 4.2.1 Linear Resonance Frequency

While the nonlinear R-P equation has no known general solution, there are solutions to the linearized form of the equation. This solution is obtained by assuming small-amplitude oscillations such that the radius can be described by,

$$ a = a_E \left[ 1 + \Re(e^{j\omega t}) \right] \quad (4.44) $$

where $a_E$ is the equilibrium radius about which the bubble oscillates, $\phi$ will in general be a complex number where the real part is the amplitude of the bubble oscillations, and $\omega$ is the frequency of the radial oscillations\[9]. Using this quantity to represent the time-varying radius, the linearized resonance frequency ($\omega$), often referred to as the natural frequency is,

$$ \omega = \sqrt{\frac{3n}{\rho a^2} \left( p - p_v + \frac{2S}{a} \right)} - \frac{2S}{\rho a^3} \quad (4.45) $$

The linear resonance frequency can be converted to units of Hertz (cycles per second) by multiplying through by $\frac{1}{2\pi}$. A plot of the linear resonance frequency for nuclei bubbles is shown in figure 4.2. This figure shows that the natural frequency of bubbles is inversely proportional to their size. Smaller nuclei bubbles will have
much higher linear resonance frequencies than larger nuclei bubbles.

Figure 4.2: Linear resonance frequency for nuclei bubbles from 1 to 500µm in radius. The linear resonance frequency is calculated from equation [4.45] under the assumption that $\rho = 1000.0 kg/m^3$, $S = 0.072 N/m$, $n = 1$, $p_v = 2340 Pa$, and $p = 101,000 Pa$.

It is important to note that the fundamental assumption in deriving the linear resonance frequency is that the oscillations of the nuclei bubble are small compared to the radial size (i.e. $\Re \phi << 1$). It is sometimes found in the literature that the linear resonance frequency, or natural frequency, is used to justify the neglect of unsteady terms in the Rayleigh-Plesset equation[21]. It is assumed that since the natural frequency is larger than the frequency of the dominant pressure fluctuations in many dynamic flowing liquids, the bubble can immediately respond to pressure changes in the liquid flow. Nuclei bubbles near the point of inception are often growing rapidly and oscillating wildly, making the assumption of small-amplitude oscillation inappropriate. Under these conditions, the nonlinearities of the problem become very important and the time period of the oscillations become amplitude dependent[29]. Under these conditions a full numerical solution of the R-P equation
is needed to determine the response to a time-varying pressure field. Later in this work, we will examine the dynamics of nuclei bubbles near the cavitation inception threshold. The assumption that the dynamic terms of the R-P equation can be neglected will be tested, along with the nonlinear response to a time-varying pressure field that is characteristic of small-scale turbulent fluctuations.

4.2.2 Radial Equilibrium Equation

The Rayleigh-Plesset equation is a second-order nonlinear partial differential equation which describes the dynamic response of a spherical bubble to a time-varying pressure field. This equation can be simplified under conditions where the nuclei bubble is exposed to a constant pressure for a period which is sufficient to allow all oscillations to be damped out. Under this condition, the dynamic terms of the R-P equation are zero. Setting the dynamic terms in the R-P equation to zero result in the equilibrium radius given by,

\[ p_0 \left( \frac{a_0}{a} \right)^{3n} - \frac{2S}{a} = p - p_v, \]  

which is a nonlinear algebraic equation. This equation is much easier to solve than the full R-P equation. However, the nonlinearity requires the use of an iterative approach such as Newton-Raphson or the bisection method. The equilibrium equation can be used to determine the size that a nuclei bubble would reach in response to a given pressure after any oscillations have been damped. This equation can also be used to determine the initial pressure inside of the bubble, \( p_0 \). This pressure is found by letting \( a = a_0 \) and setting \( p \) to the ambient pressure at which the nuclei bubble is stable \( (p_{\infty,0}) \). Equation 4.46 can be rearranged to solve directly for \( p_0 \). The result is,

\[ p_0 = p_{\infty,0} - p_v + \frac{2S}{a_0}, \]  

This equation allows the initial state of the bubble to be fully defined once the initial radius and initial ambient pressure are known.

The equilibrium equation (eq. 4.46) can also provide approximations for bubbles which are exposed to slowly changing pressure fields. If the pressure field changes
slowly enough, the dynamic terms in the R-P equation will be very small. If the dynamic terms are small enough that they are negligible with respect to the remaining terms which make up the equilibrium solution, then the equilibrium solution will provide a good approximation to the full R-P equation. The challenge is to determine a priori what constitutes an acceptably slow change in the pressure field. Without knowing this, the only way to ensure that the bubble dynamics are being represented is to numerically solve the full R-P equation.

4.2.3 Nondimensional Form: Natural

It is useful to examine the governing equations in their nondimensional form. The nondimensional form reduces the number of variables in the problem and extracts the natural variables which govern the solution to the equations. In this section, we will choose to nondimensionalize the R-P equation using the bubbles initial size (the natural length scale, hence the section title) as the length scale and derived time scales. There is more than one possibility for the derived time scale. In this section, we will present two different nondimensional forms which arise from different choices of derived time scales.

4.2.3.1 Method 1

Begin by using the nuclei bubble initial radius as the characteristic length scale. The dimensionless length scale is now defined as,

\[ \tilde{R} = \frac{a}{a_0}. \]  \hfill (4.48)

There is no obvious velocity scale to use for nondimensionalizing the radial velocity. However, the ratio of the surface tension, \( S \), to the dynamic viscosity, \( \mu \), has the units of velocity. In the absence of any other velocity scales, this ratio can provide a velocity scale to nondimensionalize the radial velocity. The result is,

\[ \tilde{R} = \frac{\dot{a}}{S/\mu}. \]  \hfill (4.49)

The time scale can be nondimensionalized by the ratio of the length scale \( (a_0) \) to the velocity scale \( (S/\mu) \). Using these quantities the dimensionless time scale is
defined as,
\[ \tilde{t} = \frac{t}{\mu a_0} \]  
(4.50)

Substituting equations 4.48 and 4.50 into the R-P equation gives,
\[
\frac{S^2}{\mu^2} \ddot{\tilde{R}} \tilde{R} + \frac{3S^2}{2\mu^2} \left( \dot{\tilde{R}} \right)^2 + \frac{4S}{a_0 \rho} \dot{\tilde{R}} + \frac{2S}{a_0 \rho} \frac{1}{\tilde{R}} - \frac{p_0}{\rho} \tilde{R}^{-3n} = \frac{p_v - p}{\rho},
\]
(4.51)

which can be simplified by multiplying through with \( \mu^2/S^2 \). The result is,
\[
\ddot{\tilde{R}} + \frac{3}{2} \left( \dot{\tilde{R}} \right)^2 + \frac{2\mu^2}{a_0 \rho S} \left( \frac{2}{\tilde{R}} + \frac{1}{\tilde{R}} \right) - \frac{p_0}{\rho S^2/\mu^2} \tilde{R}^{-3n} = \frac{p_v - p}{\rho S^2/\mu^2}.
\]
(4.52)

The equation can be cleaned up by defining some dimensionless parameters. Let \( P^* \) be a dimensionless pressure defined as,
\[
P_0^* = \frac{p_0}{\rho S^2/\mu^2},
\]
(4.53)
a natural cavitation number (\( \sigma^* \)),
\[
\sigma^* = \frac{p - p_v}{\rho S^2/\mu^2},
\]
(4.54)
and define a natural Weber number (\( We^* \)) as,
\[
We^* = \frac{\mu^2}{\rho a_0 S}.
\]
(4.55)

Using this notation, equation 4.52 becomes,
\[
\ddot{\tilde{R}} + \frac{3}{2} \left( \dot{\tilde{R}} \right)^2 + 2We^* \left( \frac{1}{\tilde{R}} \right) \left[ 2\dot{\tilde{R}} + 1 \right] - P_0^* \tilde{R}^{-3n} = -\sigma^*.
\]
(4.56)

The governing equation is now a function of only three dimensionless parameters: \( We^* \), \( P_0^* \), and \( \sigma^* \). In this form, surface tension and viscosity appear in all of the dimensionless parameters and it is difficult to isolate the effects of surface tension on the dynamics of the equation.
4.2.3.2 Method 2

The choice of dimensionless time is somewhat arbitrary. It is possible to choose an alternate dimensionless time such as\[^{73}\],

\[ \tilde{t}_{\text{alt}} = \frac{t}{\sqrt{\rho a_0^2/S}}. \]  \hspace{1cm} (4.57)

Using this dimensionless time, the dimensionless R-P equation is,

\[ \ddot{R}R + \frac{3}{2} \left( \frac{\dot{R}}{R} \right)^2 + We_{\text{alt}}^* \frac{\dot{R}}{R} + \frac{2}{R} - P_{0,\text{alt}}^* R^{-3n} = -\sigma_{\text{alt}}^* \]  \hspace{1cm} (4.58)

where,

\[ We_{\text{alt}}^* = \sqrt{\frac{16 \mu^2}{\rho a_0^2 S}}, \]  \hspace{1cm} (4.59)

\[ P_{0,\text{alt}}^* = \frac{p_0}{S/a_0}, \]  \hspace{1cm} (4.60)

and

\[ \sigma_{\text{alt}}^* = \frac{p - p_v}{S/a_0}. \]  \hspace{1cm} (4.61)

While this equation does have a different form from equation [4.56], it is still only a function of three dimensionless parameters: \( We_{\text{alt}}^* \), \( P_{0,\text{alt}}^* \), and \( \sigma_{\text{alt}}^* \). This second form is perhaps a bit more convenient because of its structure. The third term on the left-hand side of the equation is the viscous damping term. Therefore as \( We_{\text{alt}}^* \) increases, the effects of viscous damping become more pronounced. For many cavitation nuclei, the bubbles are large enough and \( We_{\text{alt}}^* \) will be small enough that the viscous damping term can be neglected. While this will affect the long term dynamics of a nuclei bubble and should not be neglected for a full cavitation study, it does simplify the mathematics and provides a very close approximation of how a nuclei bubble will respond to a perturbation in \( \sigma_{\text{alt}}^* \). Eliminating the viscous damping term reveals that this equation can be reduced to a function of only two parameters: \( P_{0,\text{alt}}^* \) and the driving dimensionless cavitation number, \( \sigma_{\text{alt}}^* \). This will be quite useful for determining when a time varying pressure field is slow enough to replace the full R-P solution with the equilibrium solution.
4.2.4 Nondimensional Form: Coupled

The previous nondimensionalization used the initial bubble size as the characteristic length scale and a derived characteristic bubble time. However, cavitation inception is a coupled phenomena where the nuclei bubble is responding to the local fluid quantities. In this section we will consider the nondimensionalization of the R-P equation using the length and time scale of the bulk liquid flow. In this form, the driving dimensionless pressure will have the same definition as the dimensionless pressure in the flow. We can also use this nondimensional form to examine how changes in the bulk flow properties will affect the parameters of the R-P equation.

The physical length scale of the bulk liquid flow is the jet diameter, $D_J$, and the characteristic physical velocity will be the jet exit velocity, $U_J$. A characteristic time can be derived from these variables as well. Using these characteristic quantities, the dimensionless variables are,

$$\tilde{R} = \frac{a}{D_J},$$

and

$$\tilde{t} = \frac{t}{D_J/U_J}.$$  (4.63)

Substituting these dimensionless variables into the R-P equation and dividing through by $U_J^2$ gives,

$$\ddot{\tilde{R}} + \frac{3}{2} \left( \dot{\tilde{R}} \right)^2 + \frac{4}{Re} \frac{\dot{R}}{R} + \frac{2}{We} \frac{\dot{R}}{R} - \left( \frac{2}{WeR_0} + \frac{\sigma_0}{2} \right) \left( \frac{\tilde{R}_0}{\tilde{R}} \right)^{3n} = -P - \frac{\sigma}{2}$$  (4.64)

where the dimensionless initial radius is defined as,

$$\tilde{R}_0 = \frac{a_0}{D_J},$$

the dimensionless pressure is defined as,

$$P = \frac{p - p_{\infty}}{\rho U_J^2},$$  (4.66)
and the cavitation number is,

$$\sigma = \frac{p_\infty - p_v}{\frac{1}{2} \rho U_j^2}. \quad (4.67)$$

The initial cavitation number is the value associated with the observation of $a_0$ and defined as,

$$\sigma_0 = \frac{p_{\infty,0} - p_v}{\rho U_j^2}, \quad (4.68)$$

where $p_{\infty,0}$ is the background pressure associated with the observation of $a_0$. The jet Reynolds number in equation 4.64 is defined as,

$$Re_f = \frac{D_J U_J}{\nu}, \quad (4.69)$$

and the Weber number, $We$, is defined as,

$$We = \frac{\rho U_j^2 D_J}{S}. \quad (4.70)$$

This is the form of the R-P equation that will be solved along with the Navier-Stokes equations for the jet flow. Notice that with this nondimensionalization, the governing equation is still a function of only two dimensionless parameters, $We$ and $Re_f$ with a source term equal to $(-P - \sigma/2)$, where $P = C_P/2$ and where $C_P$ is the coefficient of pressure within the flow defined as,

$$C_P = \frac{p - p_\infty}{\frac{1}{2} \rho U_j^2}. \quad (4.71)$$

With this method of nondimensionalization, the strength of the viscous damping in inversely proportional to the jet Reynolds number. Therefore we can expect that for very large $Re_f$ problems, viscous damping will not be significant.

### 4.3 Bubble Transport: Type-3 Scale Effects

The dynamics of nuclei bubbles are not limited to their change in volume. Nuclei bubbles are carried about the flow domain due to the bulk fluid motions. However, the velocity of a nuclei bubble is expected to be different than the velocity of the
surrounding fluid. The definitive paper on the trajectory of spherical particles at the limit of creeping flow is a paper by Maxey and Riley\cite{31}. In this paper, Maxey and Riley consider the forces on a solid sphere in the limit of creeping flow. While the motion of the nuclei bubbles may not satisfy the creeping flow assumption, their work provides a convenient starting point for the derivation of an equation of motion (EoM) for the nuclei bubbles. The equation of motion provided by Maxey and Riley for a spherical particle is,

$$m_b \frac{dV_i}{dt} = (m_b - m_f) g_i + m_f \frac{Du_i}{Dt} - \frac{1}{2} m_f \frac{d}{dt} \left\{ V_i - u_i - \frac{1}{10} a^2 \nabla^2 u_i \right\} - 6 \pi a \mu \left\{ V_i - u_i - \frac{1}{6} a^2 \nabla^2 u_i \right\} - 6 \pi a^2 \mu \int_0^t d\tau \left( \frac{d/d\tau \left\{ V_i - u_i - \frac{1}{6} a^2 \nabla^2 u_i \right\}}{\sqrt{\pi \nu (t-\tau)}} \right)$$

where $m_b$ is the mass of the particle, $m_f$ is the mass of the fluid displaced by the particle, $g_i$ is the gravitation force, $V_i$ is the velocity of the spherical particle, and $u_i$ is the velocity that the fluid would have at the center of the particle if the particle was not present. This equation is derived under the assumption of creeping flow (Reynolds number = 0). Experimental data from nuclei bubbles indicate that bubble Reynolds numbers are typically greater than 1, and often more than 10. Thus the creeping flow limit is not a realistic assumption for the motion of nuclei bubbles. Because of this, most studies of nuclei bubbles use a semi-empirical equation of motion which accounts for the change in the drag as the Reynolds number departs from zero.

It is possible to derive the semi-empirical equations of motion from the equation of Maxey and Riley. Performing this exercise shows how the individual terms in the semi-empirical equation of motion account for the underlying physics of the flow. Michaelides has pointed out that the semi-empirical equations implicitly assumes that the forces on a nuclei bubble are linearly independent, an assumption that is only appropriate in the limit of creeping flow\cite{32}. As the Reynolds number becomes order 1 or larger, the nonlinear terms in the governing equations for the fluid mechanics can no longer be neglected. On these grounds, the implicit assumptions underlying a semi-empirical EoM are theoretically precarious. However, Michaelides points out that semi-empirical EoM’s are often in very good
agreement with experimental results. This is attributed to the form of the empirical coefficients, which are obtained from experimental data and are believed to implicitly include the nonlinear effects.

To begin the derivation, assume that the effects of the Basset history term are negligible. This term makes the Maxey and Riley EoM an integro-differential equation, which is difficult to solve. Sridhar and Katz\[33\] evaluated the magnitude of the Basset history term with respect to the other terms in the EoM for a nuclei bubble and found that the relative magnitude of the Basset history term is typically small. Ruetsch and Meiburg\[42\] also showed on theoretical grounds that this term is only significant for small bubbles which are released into a flow with a velocity that does not match the local fluid velocity. In the case where the bubble is released with a significantly different velocity, the history term is significant during the initial acceleration of the nuclei bubble but decays quickly as the nuclei bubble reaches its terminal velocity relative to the flow. The term also becomes less important as the bubble Reynolds number, defined as,

$$Re_b = \frac{2a |V - u|}{\nu},$$

becomes order 1 or greater.

Other quantities which can be neglected are the Faxen terms which have the form, ($a^2 \nabla^2 u_i$). The Faxen terms were shown by Maxey and Riley\[34\] to be much smaller than the other terms in the equation, and are of secondary importance to the motion of small particles. Neglecting the Faxen and Basset history terms reduces the Maxey and Riley EoM to,

$$m_b \frac{dV_i}{dt} = (m_b - m_f) g_i + m_f \frac{Du_i}{Dt} - \frac{1}{2} m_f \frac{d}{dt} \{V_i - u_i\} - 6\pi a \mu \{V_i - u_i\}$$

If we assume the ambient flow about the bubble satisfies $Re_f >> 1$ (where $Re_f$ is the Reynolds number of the bulk liquid flow), the added mass term, $\frac{1}{2} m_f \frac{d}{dt} \{V_i - u_i\}$ can be replaced with\[34\],

$$\frac{1}{2} m_f \left( \frac{dV_i}{dt} - \frac{Du_i}{Dt} \right)$$

Furthermore, if the ambient flow about the bubble satisfies $Re_f >> 1$, then we
can replace the material derivative of the velocity field with,

\[
\frac{Du_i}{Dt} \sim -\frac{1}{\rho} \frac{dp}{dx_i}
\]

(4.76)

Substituting equations 4.75 and 4.76 into equation 4.74 gives the result,

\[
\left( m_b + \frac{1}{2} m_f \right) \frac{dV_i}{dt} = (m_b - m_f) g_i - \frac{3}{2} \frac{m_f}{\rho} \frac{dp}{dx_i} - 6\pi a \mu (V_i - u_i).
\]

(4.77)

We can now use the assumption that the bubble is always spherical to replace \( m_f \) and \( m_b \) with,

\[
m_f = \frac{4}{3} \pi a^3 \rho,
\]

(4.78)

and

\[
m_b = \frac{4}{3} \pi a^3 \rho_b
\]

(4.79)

where \( \rho \) and \( \rho_b \) are the density of the liquid outside of and the gas within the bubble, respectively. Substituting equations 4.78 and 4.79 into equation 4.77 gives,

\[
\left( \rho_b + \frac{1}{2} \rho \right) \frac{dV_i}{dt} = (\rho_b - \rho) g_i - \frac{3}{2} \frac{dp}{dx_i} - \frac{9}{2} \frac{\mu}{a^2} (V_i - u_i),
\]

(4.80)

which is the governing equation for the motion of small spherical particle, both solid and gaseous, as long as they have a no-slip boundary condition at their interface. A unique feature of nuclei bubbles is that \( \rho_b << \rho \), which allows \( (\rho_b + \frac{1}{2} \rho) \) to be replaced with \( \frac{1}{2} \rho \) and \( (\rho_b - \rho) \) to be replaced with \( -\rho \). Making these substitutions and dividing through by \( \frac{1}{2} \rho \) gives,

\[
\frac{dV_i}{dt} = -2 g_i - \frac{3}{\rho} \frac{dp}{dx_i} - \frac{9 \nu}{a^2} (V_i - u_i),
\]

(4.81)

which is an approximate equation governing the motion of spherical nuclei bubbles at the limit of creeping flow.

Equation 4.81 is essentially Newton’s first law. The left-hand side of equation 4.81 represents the acceleration of a nuclei bubble while the terms on the right-hand side represent the forces per unit mass of the accelerated fluid which act on the nuclei bubble. In order, the forces on the right-hand side represent buoyancy due to gravity, inertial effects due to pressure gradients, and the force
due to viscous drag which results when the bubble velocity is not the same as the surrounding fluid velocity. To generalize the equation, the analytic creeping drag force \((9\nu/a^2 (V_i - u_i))\) must be replaced by an empirical drag force, which for a sphere is,\[^74\]

\[
F_D = \frac{1}{2} \rho \pi a^2 C_D (V_i - u_i) |V_i - u_i| \tag{4.82}
\]

where \(C_D\) is an empirically derived drag coefficient and the velocity terms are separated for directionality. Since the mass of the bubble is negligible, inertial effects are due to the mass of the surrounding fluid which is accelerated along with the nuclei bubble. This added mass is \(\frac{1}{2} m_f\). Using this relation, the equivalent force of drag per unit mass on the spherical nuclei bubble is,

\[
\frac{F_D}{m_b} = \frac{3}{4} C_D \frac{a}{a} (V_i - u_i) |V_i - u_i| \tag{4.83}
\]

Substituting equation 4.83 for the analytical drag term in equation 4.81 gives,

\[
\frac{dV_i}{dt} = -2g_i - \frac{3}{\rho} \frac{dp}{dx_i} + \frac{3}{4} \frac{C_D}{a} (u_i - V_i) |V_i - u_i|, \tag{4.84}
\]

which is a general EoM for a spherical nuclei bubble. The range of \(Re_b\) over which this equation is appropriate depends upon the accuracy of the empirical drag coefficient, \(C_D\). White \[^75\] has published an empirical drag coefficient for a solid sphere which is reported to be accurate to within 10\% in the range of \(0 < Re_b < 2 \times 10^5\). This form for \(C_D\) is,

\[
C_D = \frac{24}{Re_b} + \frac{6}{1 + \sqrt{Re_b}} + 0.4 \tag{4.85}
\]

where \(Re_b\) is defined in equation 4.73. With the inclusion of this formula for \(C_D\), equation 4.84 becomes the EoM which will govern the motion of nuclei bubbles in the present study.

### 4.3.1 Drag Correction for Volume Variation

Johnson and Hsieh\[^36\] appear to be the first to compute the trajectory of cavitation nuclei bubbles. Their EoM was similar to equation 4.84 except that they neglected buoyancy forces and included a correction to the drag force due to the rate-of-
change in the volume of the nuclei bubble. Kornhauser showed how this drag correction term can be obtained from irrotational potential flow of an expanding spherical bubble. The form of the correction term is,

$$\frac{-3}{a} (V_i - u_i) \dot{a}.$$  \hfill (4.86)

Although presented, Johnson and Hsieh did not use this term in their work. They argued that it is only significant during violent growth or collapse, events associated with a negligible fraction of the nuclei life. Furthermore it is only a correction to the drag term, so neglecting it from the computations should result in negligible errors. Later authors have, for the most part, also neglected this term from their EoM’s. Beyond being significant for a small fraction of the nuclei bubble life, this term is strictly valid in the limit of irrotational flow about the nuclei bubble, an assumption which has already been addressed as inappropriate for this problem. Considering these issues with this correction term, it will not be included in the EoM used for the present work.

### 4.3.2 Lift Effects

One of the problems with deriving the EoM from Maxey and Riley is that there is no lateral force, or lift force, on a sphere in the limit of creeping flow. However, at finite Reynolds numbers, a lift force may be present due to bubble spin or velocity gradients in the surrounding fluid. Lift forces due to particle spin are known as Magnus effects and are typically only significant at high $Re_b$. Furthermore, this force is difficult to account for unless the dynamics of rotation for the nuclei bubble are known. Since the effects are important only at high $Re_b$ and difficult to calculate, they are almost always neglected. A second source of lift comes from the presence of a shear flow about the nuclei bubble. Auton et al. have developed an analytical model for lift force for irrotational flow and Saffman has developed an analytical model for lift from an unbounded linear shear flow. Sridhar and Katz performed a series of experiments on bubble entrained into a vortex and found that the actual lift force in a vortical flow differs from both analytical models. To improve upon calculations of the lift force, Sridhar and Katz developed an empirical lift coefficient based upon their experimental data. The
force of lift per unit added mass as developed by Sridhar and Katz has the form,

\[
\frac{3C_L}{4a} \frac{(u - V)^2}{|u - V|} \frac{(u - V) \times \omega}{|\omega|},
\]

(4.87)

where \( \omega \) is the local vorticity of the fluid, \( \epsilon_{ijk} \) is the alternating unit tensor\(^{[70]} \), and \( C_L \) is the empirical lift coefficient. The empirical lift coefficient was found to be,

\[
C_L = 0.59\alpha^{0.25},
\]

(4.88)

where \( \alpha \) is,

\[
\alpha = \frac{|\omega|a}{|u - V|}.
\]

(4.89)

From this formulation, it is apparent that the force of lift is proportional to the slip velocity to the \( \frac{7}{4} \) power, vorticity to the \( \frac{1}{4} \) power, and nuclei bubble radius to the \( -\frac{3}{4} \) power. The last term in equation (4.87) provides directionality to the lift force. Notice that the lift force is normal to both the velocity and vorticity of the local flow.

### 4.3.3 Dimensionless Equation of Motion

The full equation of motion consists of equation (4.84) with the addition of the lift term from section 4.3.2. In dimensional form, the governing EoM for spherical nuclei bubbles is,

\[
\frac{dV_i}{dt} = -2g_i - \frac{3}{\rho} \frac{dp}{dx_i} + \frac{3}{4} \frac{C_D}{a} (u_i - V_i) |V_i - u_i| + \frac{3}{4} \frac{C_L}{a} (u_i - V_i)^2 \epsilon_{ijk} \frac{(u_j - V_j) \omega_k}{|u - V| |\omega|}.
\]

(4.90)

This equation can be nondimensionalized using the dimensionless variables defined in section 4.1. For reference, the dimensionless variables are,

\[
\tilde{x}_i = \frac{x_i}{D_j},
\]

(4.91)

\[
\tilde{u}_i = \frac{u_i}{U_j},
\]

(4.92)
and

\[ \tilde{t} = \frac{t}{D_J/U_J}. \tag{4.93} \]

Substituting these into the full dimensional EoM (equation 4.90) and multiplying through by \( D_J/U_J^2 \) gives the dimensionless form of the EoM,

\[
\frac{d\tilde{V}_i}{dt} = -\frac{2}{Fr^2} g_i - 3 \frac{d\tilde{P}}{dx_i} + \frac{3 C_D}{4} \left( \tilde{u}_i - \tilde{V}_i \right) \left| \tilde{u}_i - \tilde{V}_i \right| \\
+ 3 \frac{C_L}{4} \frac{R}{\tilde{u}_i - \tilde{V}_i} \frac{\epsilon_{ijk} \left( \tilde{u}_j - \tilde{V}_j \right) \tilde{w}_k}{\left| \tilde{u} - \tilde{V} \right| \left| \tilde{w} \right|} \tag{4.94}
\]

where the Froude number is defined as,

\[ Fr = \frac{U_J}{\sqrt{D_J |g|}}. \tag{4.95} \]

The coefficient of drag is still defined as in equation 4.85 except that the bubble Reynolds number can be written as,

\[ Re_b = \frac{2a |\mathbf{u} - \mathbf{V}|}{\nu} = 2\tilde{R} \left| \tilde{\mathbf{u}} - \tilde{\mathbf{V}} \right| Re_f. \tag{4.96} \]

The coefficient of lift is also remains unchanged, defined by equation 4.88 but all of the terms in \( \alpha \) must be replaced with their dimensionless counterparts. The dimensionless form for \( \alpha \) becomes,

\[ \alpha = \frac{|\omega| a}{|u - V|} = \frac{|\tilde{\omega}| \tilde{R}}{\tilde{u} - \tilde{V}} \tag{4.97} \]

Equation 4.94 is the full dimensionless governing EoM for spherical nuclei bubbles. While included in equation 4.94 for completeness, the volume variation correction term will not be used for calculations of nuclei bubble trajectories in the present work.
4.4 Definition of Inception: Type-4 Scale Effects

The final mathematical model required for a simulation of cavitation inception is the definition of what constitutes inception. Experimentally, incipient cavitation is detected in two ways: acoustic detection and visual observation. In the first method, hydrophones are placed in the vicinity of the flow. As the background pressure in the flow is reduced, sharp “cracks” or “pops” begin to occur within specific frequency bands. As the background pressure is reduced a little more, the number of pops observed in a unit of time will increase. Eventually, the pressure will be reduced to a point where the number of pops drastically increases. This point is the acoustic definition of cavitation inception. One problem with using acoustic methods to detect cavitation inception is isolating the sound emitted from the nuclei bubbles from the noise of pumps and motors needed to create the flow. These devices often tend to mask the cavitation events, making them difficult to detect.

Mathematically, this process can be modeled by using the equations for the acoustic emissions of a spherical bubble. These emissions are given by Blake as,

\[
p_a = \frac{\rho}{x_{ob}} \left[ 2a (\dot{a})^2 + a^2 \ddot{a} \right]
\]  

(4.98)

where \( p_a \) is the acoustic pressure emitted from the nuclei bubble that would be observed at a hydrophone located a distance \( x_{ob} \) from the center of the nuclei bubble. Equation [4.98] shows that the acoustic emission of a spherical nuclei bubble are dependent upon \( \dot{a} \) and \( \ddot{a} \). Thus a static, or slowly changing, bubble is not expected to register as incipient cavitation, even if the size of the nuclei is large enough to observe directly. Also, there is some subjectivity in choosing a threshold value of \( p_a \) which defines a cavitation event. Since cavitating nuclei are associated with characteristic “pops”, or a large increase in acoustic output, it is common practice to pick a threshold value that is an order of magnitude larger than the typical acoustic emissions from nuclei bubbles in a non-cavitating simulation.

The second method of detecting incipient cavitation is through direct observation of a cavity within the flow. Experimentally this consists of having an observer watch the flow, often illuminated by a strobe light, waiting for a cavity to appear as the background pressure is reduced. What constitutes a cavity can be
somewhat subjective. One observer may declare a cavity when unsteady transient bubble are periodically observed within the flow. A second observer may wait until these short transient occurrences develop into consistent, steady cavities. These two observers will call cavitation inception at different flow conditions. To try and eliminate some of this variability, some authors report desinent cavitation conditions. Desinent cavitation conditions are those associated with the disappearance of developed cavitation as the background pressure is raised. Authors report less variability in observations of desinent cavitation[9].

Recent experiments of cavitation inception in high Reynolds number circular jets were conducted at the Garfield Thomas Water Tunnel (GTWT). This study used visual detection techniques to identify incipient cavitation[12]. Previous cavitation inception studies at this facility examined the visual detection techniques. Results indicated that cavities could be repeatable observed when they reached a diameter of $1mm[11]$. This size will be used as the definition of cavitation inception in the present work. Therefore, cavitation inception will be said to occur when a nuclei bubble reaches a physical radius of $a = 0.5mm = 500\mu m$.

In a related matter, Hsiao et al. [11] have shown through a series of numerical experiments that the two definitions of cavitation inception result in different cavitation numbers. This result is not unexpected since the two methods are based upon different nuclei dynamics. Acoustic detection depends upon rate of change in the size of a nuclei bubble, whereas visual observation depends on the physical size of a nuclei bubble. It is possible to detect cavitation using acoustic means prior to the onset of visually observable cavities if the flow conditions are sufficient to induce rapid changes in the nuclei size. This situation is referred to in the literature as “sub-visual cavitation”. Hsiao et al. concluded that it is important to use the same method of cavitation inception in numerical models if a direct comparison will be made to experimental data. The intent of the present work is to compare with the circular jet experiments conducted at the GTWT facility. Since incipient cavitation was defined visually within these experiments, a visual cavitation inception model where $a = 500\mu m$ will be used as the definition of cavitation inception for the present work.
4.5 Scaling Arguments

One of the reasons for conducting this work is to understand how the conditions which lead to cavitation inception will change between a scale model test and a prototype flow. Traditional cavitation scaling theory states that the incipient cavitation number will be the same for a model and a prototype flow if they are dynamically similar. However, experimental data appears to contradict this.

To maintain dynamic similarity, the scale model should be geometrically similar to the prototype flow. In addition, the Reynolds number of the scale-model flow must be identical to the Reynolds number of the prototype flow. Achieving this requires that the velocity of the model be equal to,

$$U_m = U_p \frac{L_p}{L_m}$$  \hspace{1cm} (4.99)

where $U_m$ and $U_p$ are the characteristic velocities of the model and prototype flow, while $L_m$ and $L_p$ are the characteristic length scales of the model and prototype flow, respectively. This relation only holds if the fluid for the model scale tests has the same kinematic viscosity as the fluid for the prototype flow.

Equation 4.64 shows that the bubble dynamics are functions of $We$ and $Re_f$. The jet Reynolds number is held constant by definition, but the jet Weber number cannot be held constant unless a fluid with different surface tension is used in the model flow. Of course this would change the relationship in equation 4.99 unless the two fluids have the same kinematic viscosity. Since it is common practice to use water for scale model tests, we will assume through the remainder of this analysis that the liquid properties are constant between the scale model and prototype liquids. To find out how the $We$ changes, equation 4.99 can be substituted into the definition of the model Reynolds number (recall that $We = \rho U^2 \infty L/\Sigma$). The results of this substitution give,

$$We_m = We_p \frac{L_p}{L_m}.$$  \hspace{1cm} (4.100)

Equation 4.100 shows that $We_m$ will be different from $We_p$ whenever the length scale of the flow is changed.

If the same liquid is used for the two flows, the initial nuclei bubble radius will
be a constant. However, the relative size of the nuclei bubbles will not be a constant between the two flows whenever the length scale is changed. The change in the relative magnitude can be found by multiplying the definition of the dimensionless initial bubble radius for the model ($\tilde{R}_{0,m} = a_0/L_m$) by $L_p/L_p$. Performing this operation and simplifying gives,

$$\tilde{R}_{0,m} = \tilde{R}_{0,p} \frac{L_p}{L_m} \quad (4.101)$$

The initial dimensionless bubble radius for the model flow will be larger than the dimensionless radius of the prototype flow. This simply means that proportionately, the initial nuclei bubbles are larger in the scale model flow.

The dimensionless initial conditions for the nuclei bubbles will also change between the scale model and prototype flows. The value of $\sigma_0$ in equation 4.64 is given by equation 4.68. Solving for the scale model value of $\sigma_0$ and substituting equation 4.99 for the model velocity give the relation,

$$\sigma_{0,m} = \sigma_{0,p} \left( \frac{L_m}{L_p} \right)^2 \quad (4.102)$$

This equation assumes that $p_{\infty,0}$ is held constant between the model and the prototype flow. There is no reason why this value must be constant between the model and prototype. There may be cases where a prototype will operate at a depth $h$ underwater. In this case, $p_{\infty,0} = -\rho gh + p_{atm}$, where $p_{atm}$ is the pressure of the atmosphere above the water surface. Assuming that the model and prototype operate at the same depth, equations 4.101 and 4.102 show that the initial dimensionless radius and initial cavitation number scale by different amounts. Because the equations are nonlinear, it is difficult to ascertain how the response of the nuclei bubble will change.

The EoM can also be examined to determine how the governing parameters change as $Re_f$ is held constant but $L_\infty$ varies. The EoM is a function of $Fr$, $C_D$, and $C_L$. Using equation 4.99 the model Froude number (given by equation 4.95) will change proportional to,

$$Fr_m = Fr_p \left( \frac{L_p}{L_m} \right)^{3/2} \quad (4.103)$$
This equation shows that the dimensionless buoyancy force will be smaller for the scale-model flow (assuming that $L_p > L_m$). Although physically the bubbles are the same size in the two flows, equation 4.103 indicates that buoyancy forces will be less important to motion of bubbles for the model flow.

The second dimensionless parameter in the equation of motion is the coefficient of drag ($C_D$). The coefficient of drag, defined by equation 4.85, is a nonlinear function of the nuclei bubble Reynolds number, $Re_b$. As a first-order approximation, $C_D$ can be set equal to only the first term of equation 4.85. Using this approximation and the initial bubble radius, $C_D$ becomes,

$$C_{D,m} \sim \frac{12}{\tilde{R}_{0,m} |\tilde{u}_m - \tilde{V}_m| Re_f^2}. \quad (4.104)$$

where the jet Reynolds number, $Re_f$, is constant assumed to be constant. This value of $C_D$ can be substituted into the term in the dimensionless EoM which represents the drag force. This substitution results in,

$$F_{D,m} \sim \frac{9}{\tilde{R}_{0,m} Re_f}. \quad (4.105)$$

Since we are also assuming that the same fluid is used for the model and prototype flows the nuclei bubbles will have the same physical radius ($a$), but a different dimensionless radius ($\tilde{R}$). Using equation 4.101, the relationship between the model and prototype dimensionless drag force is,

$$F_{D,m} \sim F_{D,p} \left(\frac{L_m}{L_p}\right)^2. \quad (4.106)$$

This shows that the relative drag force will also change when the length scale of the flow changes. It is also evident from equation 4.106 that the drag force changes faster than the buoyancy force in equation 4.103. These relations show that the relative magnitude of the terms in the EoM are also changing with length scale.

The final term which we will consider is the lift force. From the EoM, the lift
force is proportional to (assuming the bubble remains at the initial size),

$$F_L \sim \frac{3}{4} \left( \frac{0.59 \bar{\omega}^{1/4} \bar{R}_0^{1/4}}{\bar{R}_0 \left| \bar{u}_m - \bar{V}_m \right|} \right) \left( \bar{u}_i - \bar{V}_i \right) \propto 7.08 \frac{\bar{\omega}^{1/4} \left( \bar{u}_i - \bar{V}_i \right)^{3/4}}{\bar{R}_0^{3/4}} \quad (4.107)$$

While we cannot say much about how the dimensionless slip velocity will change, it is possible to comment on the dimensionless vorticity. The assumption of dynamically similar flows means that the dimensionless vorticity will be constant. Assuming that nuclei bubbles are found at the same dimensionless location within the flows and that the dimensionless slip velocities are the same, the lift force will scale by,

$$F_{L,m} \propto F_{L,p} \left( \frac{L_m}{L_p} \right)^{3/4} \quad (4.108)$$

Equation 4.108 shows that the dimensionless lift force will also change with the length scale of the flow.

These results show that even when dynamic similarity is maintained between two flows with different length scales, the equations which govern the nuclei bubbles are likely to change. While it is the view of traditional cavitation scaling theory that the cavitation number should be the same for both flows, this analysis of the dimensionless parameters which govern the nuclei bubbles indicates that bubbles will behave differently at different length scales. It is reasonable to expect that different incipient values of the cavitation number would be observed in the model and prototype flows since the dimensionless parameters which govern the nuclei bubbles cannot be held constant.
Numerical Methods

The previous chapters provided background about the physical process of cavitation inception and developed a set of mathematical equations which represent the physical processes of cavitation inception. This chapter focuses on the numerical methods used to solve these mathematical equations with a computer. The first topic to be discussed is the numerical methods used to solve the bulk fluid mechanics equations. Most of this information is documented in the CFDSHIP manual but will be briefly reviewed here for the benefit of the reader. After discussing the bulk fluid mechanics, attention will shift to the numerical methods used to solve the cavitation inception equations. Included in this discussion will be the operation and data structure of the Fortran code which was written to solve these equations. This chapter also documents the overset domain which defines the spatial locations where the governing equation are solved. Finally, a review of the computational resources required to solve the bulk fluid flow and cavitation inception governing equations will be provided.

5.1 Fluid Mechanics

The computational fluid dynamics (CFD) code used in the present study is CFDSHIP. This code was originally developed by Eric Paterson, Rob Wilson, and Fred Stern at the Iowa Institute of Hydraulic Research at The University of Iowa in Iowa City, Iowa. The code continues to be developed by a number of researchers at The University of Iowa and The Pennsylvania State University, among other
institutions. This code solves equations 4.11 and 4.12, the incompressible unsteady Navier-Stokes equations, using fully implicit finite difference methods. This system of equations forms an initial boundary value problem which requires both initial conditions and boundary conditions to obtain a solution.

CFDSHIP accepts a structured overset domain that has been preprocessed by the PEGASUS software from NASA-Ames which evaluates the overlapping meshes of the domain to determine an optimum configuration of the meshes. PEGASUS removes the coarse mesh from the solution domain when a finer mesh is present, leaving a minimum of overlap (2 mesh cells) needed to ensure a continuous solution. Information about the overlapping domain is passed to CFDSHIP via a file (the XINTOUT file) created by PEGASUS. Within this file is the information which tells CFDSHIP which points are active (field points) for the flow solution, which points interpolate the solution from surrounding meshes (fringe points), as well as points which lie outside of the flow solution entirely (hole points). The file also includes the stencils needed to interpolate data from the surrounding meshes.

Before we address the specific numerical schemes used to discretize and solve the governing equations, we must address one issue with these equations. Recall that the dimensionless governing equations are,

\[
\frac{\partial \tilde{u}_i'}{\partial \tilde{x}_i} = 0, \tag{5.1}
\]

and

\[
\frac{\partial \tilde{U}_i}{\partial t} + \bar{U}_j \frac{\partial \tilde{U}_i}{\partial \tilde{x}_j} = -\frac{\partial P}{\partial \tilde{x}_i} + \left( \frac{1}{Re_j} + \frac{1}{Re_t} \right) \frac{\partial^2 \tilde{U}_i}{\partial \tilde{x}_j \partial \tilde{x}_j} + \frac{\partial}{\partial \tilde{x}_j} \left( \frac{1}{Re_t} \right) \left[ \frac{\partial \tilde{U}_i}{\partial \tilde{x}_j} + \frac{\partial \tilde{U}_j}{\partial \tilde{x}_i} \right] \tag{5.2}
\]

A problem with these equations is that \( P \), one of the 4 unknowns, is not present in all of the equations. As such, this system of equations cannot be solved in the present form. To address this, CFDSHIP uses the pressure-implicit splitting of operators (PISO) algorithm as outlined by Issa[80]. The PISO algorithm is a predictor-corrector strategy for solving the discretized time-dependent Navier-Stokes equations in a sequential uncoupled manor. The PISO algorithm implemented in CFDSHIP consists of four steps:
1. Predictor step: the momentum equations are solved using the pressure field at the previous time step. The result is a solution for $U^*$.

2. First corrector step: a new pressure field for $P^*$ is solved using a discretized pressure-Poison equation (equation 5.3) and a corrected velocity field, $U^{**}$, is obtained by solving the momentum equations with $P^*$.

3. Second corrector step: as in step 2, the pressure-Poison is solved for $P^{**}$ and used to obtain $U^{***}$.

4. Third corrector step.

Using the PISO algorithm, equation 4.11 is not solved directly. Instead, the divergence of the momentum equation is taken and the continuity equation is used to simplify the result into a Poisson equation for pressure. This Poisson equation is,

$$\frac{\partial^2 \tilde{P}}{\partial \tilde{x}_i \partial \tilde{x}_i} = -\frac{\partial \tilde{U}_i}{\partial \tilde{x}_j} \frac{\partial \tilde{U}_j}{\partial \tilde{x}_i} + \frac{\partial}{\partial \tilde{x}_i} \left\{ \frac{1}{Re_i} \left( \frac{\partial \tilde{U}_i}{\partial \tilde{x}_j} + \frac{\partial \tilde{U}_j}{\partial \tilde{x}_i} \right) \right\},$$

(5.3)

which becomes the fourth equation in the system of governing equations, replacing the continuity equation.

CFDSHIP solves this set of equations using the finite difference method on the structured meshes. The equations are solved in the curvilinear non-orthogonal mesh coordinate system made up of coordinate directions $\xi$, $\eta$, and $\zeta$. A partial transformation is used where the independent variables are transformed and the dependent variables remain in the base Cartesian coordinate system. The temporal terms in the governing equation are discretized using the second-order backward Euler scheme,

$$\frac{\partial \phi}{\partial t} = \frac{1}{\Delta \tilde{t}} \left( \frac{3}{2} \phi^2 - 2\phi^{n-1} + \frac{1}{2} \phi^{n-2} \right),$$

(5.4)

where $\Delta \tilde{t}$ is the dimensionless time step size and the superscript “$n$” denotes the time level of the quantity $\phi$. Two previous time values of $\phi$, $\phi^{n-1}$ and $\phi^{n-2}$, must be stored to numerically calculate the temporal derivatives. All spatial derivatives are calculated implicitly, meaning they are solved at time “$n$”. The convective
terms are discretized using the second-order upwind scheme,

\[ \tilde{U}_k \frac{\partial \phi^n}{\partial \xi_k} = \frac{1}{2} \left( \tilde{U}_k + |\tilde{U}_k| \right) \left( \frac{1}{2} \phi_{i-2}^n - 2 \phi_{i-1}^n + \frac{3}{2} \phi_i^n \right) + \frac{1}{2} \left( \tilde{U}_k - |\tilde{U}_k| \right) \left( -\frac{1}{2} \phi_{i+2}^n + 2 \phi_{i+1}^n - \frac{3}{2} \phi_i^n \right) \]  

(5.5)

where \( \phi \) is a dummy variable which represents the scalar quantity of interest, the subscript \( "k" \) denotes the coordinate direction of interest and the subscript \( "i" \) denotes the nodal location of the scalar quantity with respect to the point of interest.

The viscous terms are discretized using the second-order central scheme,

\[ \frac{\partial^2 \phi^n}{\partial \xi_i^2} = \phi_{i-1}^n - 2 \phi_i^n + \phi_{i+1}^n \]  

(5.6)

in the RANS regions of the flow and a fourth-order central scheme,

\[ \frac{\partial^2 \phi^n}{\partial \xi_i^2} = -\frac{1}{12} \phi_{i+2}^n + \frac{4}{3} \phi_{i+1}^n - \frac{5}{2} \phi_i^n + \frac{4}{3} \phi_{i-1}^n - \frac{1}{12} \phi_{i-2}^n \]  

(5.7)

in the LES regions of the flow. Recall that the turbulence model requires the solution of two additional partial differential equations for \( \tilde{k} \) and \( \tilde{\omega} \). These equations are solved using first-order upwind schemes for the advective terms and second-order central schemes for the viscous diffusion terms.

The finite difference discretizations form a system of simultaneous equations which can be solved numerically. This system of simultaneous equations results in a banded matrix which is solved using the alternating direction implicit, or line ADI, technique to manipulate the system of equations into three pentadiagonal systems which are simple to solve. Alternatively, CFDSHIP also has the capability to use the PETSc package from Argonne National Laboratory\[81]\ which is a sparse linear system solver package. Unfortunately the maintainers often make changes which are not backward compatible and creating portability issues. It was also found that the PETSc solver is considerably slower than the native pentadiagonal line-ADI solver for the jet flow solution.

Since this system of governing equations represents an initial boundary-value problem, both initial conditions and boundary conditions are needed for a solution. The initial conditions for all of the dependent quantities are listed in table 5.1.
The values listed there show that the flow solution is assumed to start from rest. There are several different types of boundary conditions needed for the jet flow problem. These boundary condition types include inlet, exit, far-field, and no-slip boundary conditions which must be applied to their appropriate boundaries in the overset domain. The inlet boundary condition consists of a specified inlet velocity (a Dirichlet boundary condition) and a zero pressure gradient (a Neumann boundary condition). The turbulence quantities are also specified values, as listed in table 5.2. Note that derivatives with respect to ̂\( n \) represent the component of the derivative that is normal to the boundary of the domain.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Initial Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>̂( U_i )</td>
<td>0</td>
</tr>
<tr>
<td>̂( P )</td>
<td>0</td>
</tr>
<tr>
<td>̂( k )</td>
<td>10^{-7}</td>
</tr>
<tr>
<td>̂( ω )</td>
<td>9.0</td>
</tr>
<tr>
<td>̂( Re_t )</td>
<td>9.09 \times 10^7</td>
</tr>
</tbody>
</table>

Table 5.1: Table of initial conditions for the dependent variables in the governing equations.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Initial Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>̂( U )</td>
<td>Inlet Eq. 5.8</td>
</tr>
<tr>
<td>̂( V )</td>
<td>0</td>
</tr>
<tr>
<td>̂( W )</td>
<td>0</td>
</tr>
<tr>
<td>̂( P )</td>
<td>\frac{∂P}{∂̂( n )} = 0</td>
</tr>
<tr>
<td>̂( k )</td>
<td>10^{-7}</td>
</tr>
<tr>
<td>̂( ω )</td>
<td>9.0</td>
</tr>
<tr>
<td>̂( Re_t )</td>
<td>9.09 \times 10^7</td>
</tr>
</tbody>
</table>

Table 5.2: Inlet boundary conditions.

The inlet velocity boundary condition is initially a function of time. The specified velocity transitions smoothly from a value of zero (to match the initial conditions) to a value of 0.686. This value is the corresponding area ratio between the jet exit plane and the inlet plane of the computational domain upstream of the nozzle exit. Choosing the inlet velocity component as this ratio ensures that the average jet exit velocity will be 1. The smooth transition between these two values is defined by,

\[
\tilde{U}(\tilde{t}) = \begin{cases} 
0.686 \left[ -2 \left( \frac{\tilde{t}}{2} \right)^3 + 3 \left( \frac{\tilde{t}}{2} \right)^2 \right], & 0 \leq \tilde{t} < 2 \\
0.686, & \tilde{t} \geq 2
\end{cases} \quad (5.8)
\]
The exit boundary condition for the domain is set as the solution to the Laplace equation, \( \nabla^2 \tilde{U}_i = 0 \). All remaining dependent variables are prescribed Neumann boundary conditions as shown in table 5.3. The far-field boundary conditions are listed in table 5.4. Dirichlet boundary conditions are used for the velocity components, with the individual terms set to the free-stream velocity. For a quiescent jet, the free stream velocity is zero and thus all terms are equal to zero, however a co-flow jet will have a non-zero \( \tilde{U} \) component which is equal to the dimensionless co-flow velocity. Finally, the no-slip boundary conditions that are applied to all solid surfaces consist of Dirichlet boundary conditions for velocity and turbulence model dependent variables with a Neumann condition on pressure. The specific values of the dependent variables are listed in table 5.5. The boundary condition on \( \tilde{\omega} \) includes the parameter \( \beta \) which was defined in table 4.1 and \( \Delta \tilde{y} \) which is the distance to the first mesh point located off of the no-slip surface.
5.2 Bubble Dynamics

While many computational fluid dynamics (CFD) codes exist, there are currently no general purpose codes to perform Lagrangian analysis of cavitation inception, outside of a few research codes. To predict cavitation inception in high-Reynolds number circular jets, a Lagrangian cavitation inception code was developed which predicts cavitation inception in a fluid flow obtained from a separate CFD code. While the code was written to solve the problem of cavitation inception in a high-Reynolds number circular jet, it has been written in a general fashion so as to be easily modified to solve other classes of problems (solid particle, neutrally buoyant tracer particles) with flow solutions from arbitrary flow solvers (general unstructured or structured overset meshes). This section documents the specific numerical methods needed to solve the problem of cavitation inception in circular jets using flow solutions obtained from CFDSHIP and where appropriate, comments are made regarding other capabilities which have been included in the code.

5.2.1 Code and data structures

The bubble dynamics code is written entirely in Fortran 90. It makes use of dynamic data structures available in Fortran 90 as well as message-passing interface (MPI) libraries for parallel computation. While the data input and output are based upon CFDSHIP and Ensight file formats, the code is written to accept general unstructured data. New input and output subroutines would be needed to handle these based upon the flow solver and visualization packages that are desired. However, the code is written in such a fashion that these modifications can be made quickly. Implementation of general unstructured data would also require the creation of new subroutines for the specific unstructured elements which are required (pyramid, tet, etc.). Placeholder for these subroutine calls already exist within the code and the data structure for these general elements is already implemented.

To handle general unstructured and structured domains, all meshes are treated as unstructured. Since CFDSHIP is used to obtain flow solutions, only structured domains need to be handled for the present jet problem. Structured domains are treated as a general unstructured meshes which consists entirely of 6-sided hex
elements. While file input/output for general unstructured meshes is not presently implemented in the code, the data structure within the code allows for general meshes which can consist of tetrahedrons, prisms, wedges, and hexahedrons. To allow for these different elements, the code allocates an array for each element type which has a length equal to the number of the corresponding element types in the mesh. Dynamic data types then allow an integer array, named \texttt{node\_map( )}, to be associated with each entry in the 1-dimensional element array. The size of \texttt{node\_map( )} is equal to the number of nodes which define the corresponding element type. This array is then filled with the corresponding node numbers associated with the nodes that make up the element. For example, hex elements are stored in an array with the name \texttt{hex}. Each entry in \texttt{hex} is associated with array \texttt{node\_map} which has a size of 8, corresponding to the 8 nodal values which make up a hex element stored in the order shown in figure 5.1. Using derived data types, the nodes associated with a specific hex element are accessed using the variable, \texttt{hex(213)%node\_map(3)}, which would return the node number associated with the third node of the 213\textsuperscript{th} hex element.

![Figure 5.1: Order of node numbering for a hex element.](image)

The fluid solutions from CFDSHIP are stored in Ensight Gold format for the variables $\tilde{U}, \tilde{V}, \tilde{W}, \tilde{P}$, and $\tilde{k}$. The values associated with these variables, along with the $\tilde{x}, \tilde{y}, \tilde{z}$ data and the \texttt{iblank} value (overset structured meshes only) for each node are stored using derived data types. A 1-dimensional array, named \texttt{node}, is declared with a size equal to the total number of nodes in the computational domain. Using derived data types, the variable associated with each node can be accessed by calling \texttt{node(1054)%u}, which would return the value of the
\( \tilde{U} \) component of the velocity vector associated with node number 1054. Derived data types allow access to the flow variables associated with a node in an element by calling \( \text{node(hex(213)%node_map(3))\%u} \), which would return the \( \tilde{U} \) component of velocity for the third node associated with the 213\textsuperscript{th} hex element in the computational domain.

The data associated with the Lagrangian nuclei bubbles (or Lagrangian particle) is also stored using derived data types. The values associated with the nuclei bubbles are associated with a vector called \( lgn \), which has a size equal to the number of nuclei bubbles. The quantities associated with a Lagrangian bubble are stored in variables with the form, \( lgn(i)\%r \), which returns the radius associated with the \( i \)\textsuperscript{th} Lagrangian bubble. Other variables associated with the Lagrangian bubble array are, location \((x,y,z)\), velocity \((u,v,w)\), radius \((r)\), radial velocity \((dr)\), radial acceleration\((d2dr2)\), initial radius \((r0)\), and initial gas pressure \((pr_g0)\). Along with these variables, the properties of the flow solution at the location of the Lagrangian bubble are also stored. The properties include, local velocity \((u_{loc}, v_{loc}, w_{loc})\), the local pressure \((pr_{loc})\), the pressure gradient \((\text{grad}_p(i))\), the velocity gradients \((\text{grad}_u(i), \text{grad}_v(i), \text{grad}_w(i))\), and the vorticity \((\omega(i))\). The variable \( i \) associated with the vector quantities specifies the directionality of the component and therefore has values of 1, 2, or 3. The derived data structure also includes information about the element which the Lagrangian bubble lies within. This element information includes the element type \((\text{element\_type})\), and the element number \((\text{element\_num})\).

The layout and operation of the bubble dynamics code is shown in figure 5.2. The first action of the code is to read the input files to determine the operating parameters of the code. The input files include \texttt{lagrangian.nml} which dictates the options for executing the code along with the operating conditions, \texttt{size\_distribution.dat} which lists the number and size of the nuclei bubbles which will be randomly distributed within the inlet domain, and \texttt{filename.rsto} which is a restart file for nuclei bubbles which will be restarted from a previous simulation. The first two files \( \texttt{lagrangian.nml, size\_distribution.dat} \) are ASCII text files while the restart file is stored in binary form.

The next action the code takes is reading in the geometry data from a file. While the code is written in a general fashion to accept both structured and un-
structured data, the current implementation of the code can only read structured
Figure 5.2 (continued)
mesh data stored in ASCII Plot3D format. Placeholders exist to accept subroutines for unstructured mesh files, should they become necessary in the future. As the structured mesh is read into memory, the data is converted into unstructured hex elements. A structured mesh with \( n_i \) points in the i-direction, \( n_j \) points in the j-direction, and \( n_k \) points in the k-direction will result in \( (n_i-1) \times (n_j-1) \times (n_k-1) \) structured hex elements. The code also reads the PEGASUS XINTOUT file to determine which nodes associated with the mesh have been blanked out. An element must include at least 1 valid field point to be added to the list of elements for the subsequent bubble dynamics calculations. Any element which consists entirely of hole or fringe elements will be ignored.

After reading the overset domain data, the Lagrangian bubbles are distributed into the flow. New bubbles are declared in \texttt{size\_distribution.dat} and are randomly placed into a volume, called the inlet domain, that is defined by the user. The inlet domain can be a rectangular box, or a circular cylinder oriented along the \( x \)-axis as defined in \texttt{lagrangian.nml}. The Lagrangian bubbles are advected with a dimensionless velocity of 1.0 while inside of the inlet domain and at the end of each time step, a check will be performed to see if the bubble has left the inlet domain. If it has left the inlet domain, the bubble will be subsequently governed by the EoM and R-P equations (if applicable). After all of the bubbles declared in \texttt{size\_distribution} have been distributed, the code checks for the existence of a restart file. If found, the bubbles declared in this file are added to the list of nuclei bubbles.

At this point the code has all of the information that is needed to begin the calculations. The code begins a time stepping loop to advect the Lagrangian bubbles with the available bulk flow data. On the first time step, the code loads data from both the first and second flow time steps so that the solutions can be linearly interpolated in time. The flow solution data is assumed to be in Ensight Gold binary format data files. Velocity and pressure data are required variables whereas turbulent kinetic energy data is input only if available. When the turbulent kinetic energy data is not available, a warning is written to standard output and the code will continue under the assumption that the correction to the pressure variable due to turbulent kinetic energy is zero.

After reading the flow solution data into memory, the code loops over each
Lagrangian bubble and perform sub-time step calculations. Bubbles flagged as being within the inlet domain will be advected a distance equal to the dimensionless velocity (1.0) times the subiteration time step size. If the bubble is not flagged as being in the inlet domain, a search is performed to find which element the bubble is within. If no element can be found, the bubble is deactivated. Otherwise, the flow data from the element is interpolated to the location where the bubble is found and the governing equations for the bubble are numerically integrated in time over the sub-time step. The code provides the option of integrating the governing equation using either a first-order Euler method or the fourth-order Runge-Kutta method.

Once the nuclei bubble dynamics have been integrated over all sub-time steps, the state of the nuclei bubble is written to disk. The output is written in Ensight Gold Measured Particle format which consists of one file per variable per time step. A restart file for the bubble data is also written to disk at this time if the user has specified this feature. When the file output is complete, the code loops to the next flow time step and repeats the process just described until all time steps have been used.

### 5.2.2 Search and Interpolation Algorithms

A key function of the Lagrangian bubble dynamics code is to determine which cell a Lagrangian bubble is within and then to interpolate the fluid properties from the element nodes to the point where the bubble is located. This process is handled by subroutine `locate_particle`, which first checks to see if an element type and element number are already associated with the bubble. If an element type and element number are associated, a detailed search is performed to determine if the bubble is still within that element. If the bubble is not within that element, or if no element type or element number were associated with the bubble, the code searches over all elements to determine which cell the bubble is within. The algorithm for checking to see if a bubble is within an element consists of several steps. The first step is to gather the geometric data about the nodes which make up the element. Then a quick check is performed to determine if the bubble is within the extrema of the element. If the bubble is within the extrema, a detailed search is performed to determine if the bubble is really within the element, or just within the extrema.
as illustrated in figure 5.3. The detailed search consists of finding the coordinate value of the bubble within an element-based coordinate system. This coordinate system has coordinate directions of $\xi, \eta,$ and $\zeta$ with the nodal coordinate values listed in figure 5.4.

![Figure 5.3: Example of a bubble that is within the extrema of an element, but not within the element itself.](image)

![Figure 5.4: Coordinate values for the nodes as defined for the element coordinate system.](image)

Using linear interpolation, the physical coordinates of any point within a hex element can be found if the coordinates of the node points are known. The physical coordinates will be given by,

$$\tilde{x}_i = \sum_{n=1}^{8} \psi_n \tilde{x}(i, n)$$

(5.9)

where $\psi_n$ is the shape function of the element type, which for a hex element is,

$$\psi_n = \frac{1}{8} (1 + \xi \xi_n) (1 + \eta \eta_n) (1 + \zeta \zeta_n).$$

(5.10)

The variable, $\tilde{x}_i$, is the physical coordinate of the point, the variables $\xi_n, \eta_n,$ and $\zeta_n$ are the coordinate values associated with nodes of the hex element, and $\tilde{x}(i, n)$
is the physical location of node $n$. The goal is to find the $\xi$, $\eta$, and $\zeta$ values such that $\tilde{x}_i$ is the same as $\tilde{\tilde{x}}_i(i, p)$, where $\tilde{\tilde{x}}_i(i, p)$ is the coordinate value in the $i$-direction for the Lagrangian bubble. This value can be solved in an iterative fashion using Newton’s method to find the root of the equation $\tilde{x}_i - \tilde{\tilde{x}}_i(i, p) = 0$. The iteration starts with an initial guess ($\xi_i^k$) and solves for a corrected value ($\xi_i^{k+1}$) using,

$$\xi_i^{k+1} = \xi_i^k + h_i$$

(5.11)

where $h_i$ is defined as,

$$h_i = -\frac{\tilde{\tilde{x}}_i^k - \tilde{x}(i, p)}{J^k}.$$ 

(5.12)

The variable $J$ is the $3 \times 3$ Jacobian matrix defined as,

$$J_{ij} = \frac{\partial \tilde{x}_i}{\partial \xi_j}$$

(5.13)

The correction term, $h_i$, can be found by using Gauss elimination to solve,

$$J_{ij}^k h_i = \tilde{\tilde{x}}_i(i, p) - \tilde{x}_i^k$$

(5.14)

for the unknown quantity, $h_i$. Once $h_i$ is known, the corrected values of $\xi_i$ is given by equation 5.11. The process is then repeated until $\sqrt{(\xi_i^{k+1} - \xi_i^k)^2} < 1.0 \times 10^{-6}$.

At this point, the code knows the value of $\xi$, $\eta$, and $\zeta$ to within about $1.0 \times 10^{-6}$. Theoretically, the bubble lies within the element if $-1.0 \leq \xi \leq 1.0$. However, since there is some error in the numerical calculation, the code considers the bubble to be within the element if,

$$-1.0 - 1.0 \times 10^{-6} \leq \xi_i \leq 1.0 + 1.0 \times 10^{-6}$$

(5.15)

Once an element is found in which the bubble is within, the local coordinate system may be used to linearly interpolate the flow solution from the nodes of the element to the location of the bubble. The local value of any fluid variable is

$$\phi = \sum_{n=1}^{8} \frac{1}{8} (1 + \xi \xi_n) (1 + \eta \eta_n) (1 + \zeta \zeta_n) \phi_n$$

(5.16)
where $\phi_n$ is some known quantity that is associated with node $n$ and $\phi$ is the value of the known quantity at the location of the bubble. This process is used to find the local fluid velocity components and pressure at the point where the bubble is found. The EoM also requires gradients of pressure and velocity. These quantities are found by solving,
\[
\nabla \phi = \sum_{n=1}^{8} (J^T)^{-1} \nabla \psi_n \phi_n. \tag{5.17}
\]

where $J$ is the Jacobian defined in equation \ref{eq:5.13}.

One final interpolation process is needed to obtain the value of the fluid variables associated with the sub-time steps. The value of the variables at each node of the element are linearly interpolated from the previous and future fluid time step values. This provides the bubble with a continuous solution in time for the local fluid variables.

### 5.2.3 Numerical Solution of the Rayleigh-Plesset Equation

The code provides two methods for solving the R-P equation. The first method, which will not be discussed here, is the first-order explicit Euler method. This method is straightforward to program but only provided first-order accuracy. Pseudo code for the Euler method can be found in most textbooks on numerical methods for scientists and engineers\cite{82}. The second method for solving the R-P equation is the fourth-order Runge-Kutta method. This method is preferred over the Euler method as it allows larger time steps and provides higher order accuracy. Pseudo code for the Runge-Kutta method implemented in this work can be found in Cheney and Kincaid\cite{82}.

The fourth-order Runge-Kutta method (RK4) can be used to solve initial value problems. A slight complication with the R-P equation is that it is a second-order ODE, whereas RK4 can only handle single derivatives. This can be addressed by solving for two separate equations,
\[
\frac{d}{dt} (\ln(m)\%r) = \ln(m)\%dr \tag{5.18}
\]

and
\[
\frac{d}{dt} (\ln(m)\%dr) = \ln(m)\%d2rdt2 \tag{5.19}
\]
where \( lgn(m)\%dr \) is the time derivative of the radius \( \dot{R} \) for Lagrangian bubble number \( m \) and \( lgn(m)\%d2rdt2 \) is the second derivative of the radius with respect to time \( \ddot{R} \) for Lagrangian bubble \( m \). To begin the RK4 algorithm, 2 initial conditions are required, one for \( lgn(m)\%r \) and one for \( lgn(m)\%dr \). The initial condition for \( lgn(m)\%r \) is the initial radius for the bubble, \( \tilde{R}_0 \), while the initial radial velocity is assumed to be zero. With these values, the following steps are the RK4 algorithm.

\[
lgn_r\_store = lgn(m)\%r \ (r_0 \text{ for the first time step})
\]

\[
lgn_dr\_store = lgn(m)\%dr \ (0 \text{ for the first time step})
\]

\[
K_r(1) = \Delta t \ ( lgn(m)\%dr )
\]

\[
K_{dr}(1) = \Delta t \ ( RP(m) )
\]

where \( RP(m) \) is a function call that returns the solution to \( \ddot{R} \) in equation 4.64.

\[
lgn(m)\%r = lgn_r\_store + \frac{1}{2}K_r(1)
\]

\[
lgn(m)\%dr = lgn_dr\_store + \frac{1}{2}K_{dr}(1)
\]

\[
K_r(2) = \Delta t \ ( lgn(m)\%dr )
\]

\[
K_{dr}(2) = \Delta t \ ( RP(m) )
\]

\[
lgn(m)\%r = lgn_r\_store + \frac{1}{2}K_r(2)
\]

\[
lgn(m)\%dr = lgn_dr\_store + \frac{1}{2}K_{dr}(2)
\]

\[
K_r(3) = \Delta t \ ( lgn(m)\%dr )
\]

\[
K_{dr}(3) = \Delta t \ ( RP(m) )
\]

\[
lgn(m)\%r = lgn_r\_store + K_r(3)
\]

\[
lgn(m)\%dr = lgn_dr\_store + K_{dr}(3)
\]

\[
K_r(4) = \Delta t \ ( lgn(m)\%dr )
\]

\[
K_{dr}(4) = \Delta t \ ( RP(m) )
\]
The final values of $lgn(m)\%r$, $lgn(m)\%dr$, and $lgn(m)\%d2rdt2$ are the fourth-order accurate values at the new time step.

### 5.2.4 Numerical Solution of the Equation of Motion

The same RK4 algorithm is used to solve the EoM, which is a second-order ODE for position. This problem is also an initial value problem which will require two initial conditions. The following section of code only shows the components associated with the $x$-direction as the $y$ and $z$-directions are solved in the exact same manner.

\[
lgn_x = lgn_x_{\text{store}} + \frac{1}{2} K_x(1)
\]
\[
lgn_u = lgn_u_{\text{store}} + \frac{1}{2} K_u(1)
\]
\[
K_x(2) = \Delta t \times lgn(m)\%u
\]
\[
K_u(2) = \Delta t \times \text{lagrangian_EoM}(m)
\]

where $\text{lagrangian_EoM}(m)$ is a function call that returns the solution to $d\tilde{V}/d\tilde{t}$ from equation 4.94 without including the volume variation term.

\[
lgn(m)\%r = lgn_r_{\text{store}} + \frac{1}{6} [K_r(1) + 2 (K_r(2) + K_r(3)) + K_r(4)]
\]
\[
lgn(m)\%dr = lgn_dr_{\text{store}} + \frac{1}{6} [K_{dr}(1) + 2 (K_{dr}(2) + K_{dr}(3)) + K_{dr}(4)]
\]
\[
lgn(m)\%d2rdt2 = RP(m)
\]
\[ K_u(3) = \Delta t \times \text{lagrangian}_\text{EoM}(m) \]

\[ lgn(m)\%x = lgn_x\_store + K_x(3) \]
\[ lgn(m)\%u = lgn_u\_store + K_u(3) \]

\[ K_x(4) = \Delta t \times lgn(m)\%u \]
\[ K_u(4) = \Delta t \times \text{lagrangian}_\text{EoM}(m) \]

\[ lgn(m)\%x = lgn_x\_store + \frac{1}{6} [K_x(1) + 2(K_x(2) + K_x(3)) + K_x(4)] \]
\[ lgn(m)\%u = lgn_u\_store + \frac{1}{6} [K_u(1) + 2(K_u(2) + K_u(3)) + K_u(4)] \]

### 5.2.5 Input Files

There are 3 main input files for the Lagrangian bubble dynamics code, two which define the bubbles which will be released into the simulation and one to input the control parameters to obtain the desired behavior from the simulation. The file which inputs the control parameters is the most important file to the operation of the code. This file is named \texttt{lagrangian.nml}. This is a Fortran namelist file which provides the opportunity to input parameters which will govern the operation of the Lagrangian bubble dynamics simulation. The namelist file accepts 5 namelist groups: \texttt{control}, \texttt{input\_files}, \texttt{flow\_properties}, \texttt{bubble\_properties}, and \texttt{inlet\_domain}.

The \texttt{control} namelist accepts variables which govern the operation of the code. These variables are listed in table 5.6 along with the default values which are assumed if the variables are not set in the namelist file. A brief description of what that variable controls is also provided. The \texttt{input\_files} namelist provides the flow solver with the file names that will be needed to load the mesh, flow solutions, and any previous bubble data. The variables which are required in this section are listed in table 5.7. The \texttt{flow\_properties} namelist provides the flow solver with the values of the parameters used to create the flow solution. It also contains parameters which describe some physical aspects of the flow needed to nondimensionalize the Lagrangian bubble properties. The variables which can be defined in this section are listed in table 5.8. The \texttt{bubble\_properties} namelist
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>its</td>
<td>Integer</td>
<td>1</td>
<td>Starting time step number</td>
</tr>
<tr>
<td>ite</td>
<td>Integer</td>
<td>1</td>
<td>Ending time step number</td>
</tr>
<tr>
<td>it_increment</td>
<td>Integer</td>
<td>1</td>
<td>Incremental value to advance the time step</td>
</tr>
<tr>
<td>delt</td>
<td>Double</td>
<td>0.01</td>
<td>Flow solver time step size (dimensionless)</td>
</tr>
<tr>
<td>n_sub_tsteps</td>
<td>Integer</td>
<td>$1 \times 10^5$</td>
<td>Number of subiteration time steps</td>
</tr>
<tr>
<td>input_type</td>
<td>Character</td>
<td>“plot3d”</td>
<td>Format of input mesh data</td>
</tr>
<tr>
<td>advection_scheme</td>
<td>Character</td>
<td>“rk4”</td>
<td>Method of numerical integration (Options: “rk4” or “euler”)</td>
</tr>
<tr>
<td>bubble_dynamics</td>
<td>Logical</td>
<td>.FALSE.</td>
<td>Command to turn on or off solution of the Rayleigh-Plesset equation</td>
</tr>
<tr>
<td>pathlines</td>
<td>Logical</td>
<td>.FALSE.</td>
<td>Option to advect the bubbles as massless tracers</td>
</tr>
<tr>
<td>stead_flow</td>
<td>Logical</td>
<td>.FALSE.</td>
<td>Option to allow for steady state flow solutions</td>
</tr>
<tr>
<td>make_restart_file</td>
<td>Logical</td>
<td>.FALSE.</td>
<td>Option to write a restart file at the end of the simulation</td>
</tr>
</tbody>
</table>

Table 5.6: Input parameters available in the control namelist in lagrangian.nml. These parameters select the numerical methods and operating parameters of the numerical simulation.

provides the flow solver with the values of the parameters needed to calculate the dynamic response of the nuclei bubble using the R-P equation. The variables which need to be defined are listed in table 5.9. Finally, the inlet_domain namelist is used to specify the parameters which define the inlet box. The nuclei bubbles from the file “size_distribution.dat” are randomly distributed throughout the inlet box. The variables which can be set in this namelist are listed in table 5.10 and graphically represented in figure 5.5.

5.2.6 Output Files and Post Processing

The Lagrangian bubble dynamics code was designed to work with Ensight software from CEI Incorporated. The Lagrangian bubble information is saved at each flow time step as Ensight Gold “measured particle” data. It would be ideal to have...
Table 5.7: Input files that are required by the code to load the fluid dynamics solutions.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>grid_fname</td>
<td>Character</td>
<td></td>
<td>File which contains the overset domain</td>
</tr>
<tr>
<td>xintout_fname</td>
<td>Character</td>
<td></td>
<td>File which contains the XINTOUT data for the overset domain</td>
</tr>
<tr>
<td>root_filenamei</td>
<td>Character</td>
<td></td>
<td>Base filename for the Ensight Gold files which contain the transient velocity, pressure, and turbulent kinetic energy data</td>
</tr>
<tr>
<td>root_filenameo</td>
<td>Character</td>
<td></td>
<td>Base filename for the Ensight Gold files which will be written to record the Lagrangian bubble results</td>
</tr>
<tr>
<td>restart_filename</td>
<td>Character</td>
<td></td>
<td>Name of the restart file to be read into the code</td>
</tr>
</tbody>
</table>

Table 5.8: Parameters which describe the fluid flow. These properties are set in the flow_properties namelist in lagrangian.nml.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re</td>
<td>Double</td>
<td>10,000</td>
<td>Reynolds number associated with the fluid dynamics solutions</td>
</tr>
<tr>
<td>L_dim</td>
<td>Double</td>
<td>1.0</td>
<td>Physical reference length of the fluid dynamics solution in meters</td>
</tr>
<tr>
<td>kinematic_viscosity</td>
<td>Double</td>
<td>$1.17 \times 10^{-6}$</td>
<td>Kinematic viscosity of the bulk liquid</td>
</tr>
<tr>
<td>density</td>
<td>Double</td>
<td>1000.0</td>
<td>Density of the bulk liquid in kilograms per cubic meter</td>
</tr>
<tr>
<td>gravity_dir</td>
<td>Integer</td>
<td>-2</td>
<td>Direction in which gravity points (1=x-direction, 2=y-direction, 3=z-direction)</td>
</tr>
</tbody>
</table>

The sub-time step data stored as well, however, due to the way that Ensight stores transient data, output files would have to be created for each sub-time step as well. The resulting amount of data that would be written to disk would increase by a factor of the number of sub-time steps. Considering that the number of sub-time steps is typically greater than 10,000, this would be a tremendous increase in the number of files and time spent writing data to disk. Since the purpose of this code is to document cavitation events, it is not necessary to know the properties of the
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>sigma</td>
<td>Double</td>
<td>100</td>
<td>Cavitation number of the simulation</td>
</tr>
<tr>
<td>sigma0</td>
<td>Double</td>
<td>Calculated</td>
<td>Initial cavitation number associated with the measurement of $\tilde{R}_0$. If not specified, this value is calculated assuming that measurements were made at standard atmospheric pressure</td>
</tr>
<tr>
<td>surf_tension</td>
<td>Double</td>
<td>0.072</td>
<td>Surface tension in Newtons per meter</td>
</tr>
<tr>
<td>kgas</td>
<td>Double</td>
<td>1.4</td>
<td>Polytropic exponent for compression and expansion of the gas within the nuclei bubble</td>
</tr>
</tbody>
</table>

Table 5.9: Parameters used to solve the Rayleigh-Plesset equation. These properties are set in the bubble_properties namelist in lagrangian.nml.

Lagrangian bubble at each sub-time step. Obtaining data at the flow time steps will be sufficient to determine if cavitation has occurred within the flow time step.

After the first time step is computed, two initial files are written to disk. The first file is the Ensight Gold case file which defines all of the parameters which Ensight will need to visualize both the transient flow solution and the transient Lagrangian bubble data. The second file is the Ensight Geometry file which saves the overset domain data and iblank values. Currently, the code can only handle structured overset data. If general unstructured data is needed, the file “initialize_ensight.F90” will require modifications to write a general unstructured Ensight
geometry file.

After each time step is complete, the Lagrangian data is written to disk in En-\sight Gold Measured Particle format. The files written to disk at each time step include a measured particle geometry file named, “root_fnameo_geo.lgn.XXXXX”, where “root_fnameo” is specified by the user in “lagrangian.nml” and the file extension, XXXXX, is the time step number which corresponds to the data. Other files include the bubble velocity components (“root_fnameo_lgn_vel.vec.XXXXX”), the local flow velocity components (“root_fnameo_lgn_locvel.vec.XXXXX”), the local pressure values (“root_fnameo_lgn_pr.scl.XXXXX”), the bubble radius (“root_fnameo_lgn_r.scl.XXXXX”), the bubble radial velocity (“root_fnameo_lgn_dr.scl.XXXXX”), and the bubble radial acceleration (“root_fnameo_lgn_d2rdt2.scl.XXXXX”). The user should not have to make any modifications or corrections to view this data in Ensight8 from CEI.

\section{Domain Decomposition and Overset Grid Generation}

CFDSHIP is a finite difference code which requires structured meshes. To resolve complicated geometries, CFDSHIP also accepts overset domain decomposition, which allows the domain to be resolved by overlapping structured meshes. This section will focus on providing a very brief overview of the process of generating overset domains, followed by a review of the meshes which are used to obtain the computational results discussed in the following chapters.

\subsection{Overset domain Generation}

The first step in the process of generating structured overset domains for CFDSHIP is to generate a series of overlapping structured meshes that resolve the geometry of the problem. Gridgen software from Pointwise, Inc. was exclusively used to generate the meshes for the present work. In the process of obtaining the meshes for this problem, many different topologies were tried. Some topologies consisted entirely of overlapping boundaries while others consisted of a mix of overlapping and block-to-block boundaries. An important consideration in the development
of the overlapping domains is that CFDSHIP requires double-fringe holes and boundaries. A double fringe boundary consists of a field point (a node where the variables are determined by solution of the N-S equations) with at least two fringe points in each direction. The fringe points are nodes in which the value of the flow variables is interpolated from field points in a neighboring mesh, instead of by solution of the N-S equations. All field points must have at least two other field points or fringe points in each direction to provide information to the 5-point stencils used for the finite difference representation of the governing equations.

An advantage to the block-to-block boundaries is that the meshes can be directly used by CFDSHIP without using PEGASUS software to calculate the holes and double fringe boundaries. However, the shared boundary of the two blocks must have the same number and distribution of nodes. This can lead to an inefficient distribution of mesh points and an overly expensive computation. Another disadvantage to using block-to-block boundaries in CFDSHIP is that the code resolves block-to-block boundaries with first-order accurate finite difference representations of the partial derivatives. Thus, while the remainder of the flow-field is resolved with second-order accurate finite-difference methods, these boundaries will tend to decrease the overall order of accuracy.

To get around these limitations, full overset boundaries may be used. However, the process of obtaining the interpolation files from PEGASUS can be quite time consuming. Quite often, the first few overset domains will cause problems with PEGASUS returning orphan points (points which don’t have enough neighbors or a valid interpolation stencil). Some orphan points can be fixed by changing configurations in the PEGASUS configuration files while other orphan points must be addressed by modifying the meshes and then iterating. Regardless, the iterative process of fixing orphan points within PEGASUS can be quite time consuming.

Once a suitable overset domain is obtained, appropriate boundary conditions are applied and the overset domain is used by CFDSHIP to obtain a solution. However, CFDSHIP may not converge to a solution with all overset domains. In some cases solutions can be obtained by modifying the initial conditions or boundary conditions, but other times the only way to get a solution is to modify the overset domain and begin the PEGASUS process all over again, further increasing the time required to get a solution with CFDSHIP. In general, this iterative process
of generating overset domains which are suitable for PEGASUS and CFDSHIP can be quite time consuming, requiring a good deal of practice and experience.

The next section describes the overset domain which is used to obtain the computational results which will be discussed in detail in later chapters. This mesh is the product of one year’s worth of work developing meshes that were acceptable to both PEGASUS and CFDSHIP. At least 8 different overset domains were developed along the way, with many smaller iterations on each of these overset domains needed to eliminate orphan points from the PEGASUS output. In an effort to keep this chapter a reasonable size, these overset domains will not be discussed in detail. We will note thought that the final overset domain discussed in the next section incorporates many features that were learned in the iterative process of developing these earlier computational domains.

5.3.2 Computational Overset Domain

The topology of the final overset domain is a hybrid block-to-block and overset boundary domain which required 69 meshes to resolve with proper load balancing. This section will attempt to describe the topology of the meshes and the resolution of the flow field. Outlines of the 69 meshes which make up the computational domain are illustrated in figure 5.6. It is clear from this figure that the system of overset meshes is quite complicated when presented on a 2-dimensional medium. In an attempt to reduce the complexity, the meshes will be presented in small groups which make up limited regions of the domain. At the end of this section the reader should be able to mentally assemble the 3-dimensional spatial relationship between the meshes from the following series of 2-dimensional pictures.

The far-field region of the flow is resolved using a cylindrical mesh and a rectangular far-field core mesh. Both are illustrated in figure 5.7. Only the back face and axial split surface are visualized for the cylindrical far-field mesh (surfaces shown in blue). The remaining surfaces have been turned off to allow visibility of the far-field core mesh (red) and wire-frame representations of the remaining 67 meshes (gray) in the domain. The axial split of the cylindrical mesh is the only block-to-block interface for either of these two meshes. The cylindrical mesh contains 151,725 node points with 85 points in the axial direction, 35 points in the
radial direction, and 51 points about the circumference. The rectangular far-field core consists of 150,000 node points, 60 nodes in the direction of the cylindrical mesh axis by 50 nodes each, in the remaining two directions.

The central core of the cylindrical far-field mesh upstream of the far-field core mesh is filled with 67 meshes used to resolve the flow through, around, and downstream of the nozzle. The exterior surface of the nozzle is resolved using 8 meshes which are grown radially from the surface with block-to-block boundaries. These meshes are shown in figure [5.8]. All eight meshes have 80 nodes in the direction of the nozzle axis, 40 nodes normal to the nozzle surface, and 52 nodes circumferentially for a total of 166,400 nodes each. The near-wall spacing for the nodes in the wall normal direction are set to $4.0^{-5}$ to ensure that the $y^+$ values at the wall are less than 1 for a flow with a Reynolds number of 500,000.

The topology of the meshes which make up the inner surface of the nozzle
are very similar. These meshes are shown in figure 5.9 and have dimensions of 75 nodes in the nozzle axis direction, 40 nodes normal to the nozzle wall, and 52 nodes circumferentially for a total of 156,000 node points. The near-wall spacing is identical to the outer surface values, $4.0^{-5}$. Figure 5.9 also includes a magnified view which shows that the meshes are clustered about the nozzle exit in the axial direction. This clustering is needed to maintain a similar cell size with the meshes needed to resolve the surface of the nozzle lip and the downstream separated shear layer. The mesh spacing at the nozzle exit in the axial direction is set to $1.0 \times 10^{-4}$.

The meshes that resolve the surface of the nozzle lip and the separated shear layer which will leave the nozzle are shown in figure 5.10. This region of the
flow is resolved with 32 meshes which are refined to capture both inner and outer separated shear layers and the boundary layers on the nozzle lip. All of these
blocks share block-to-block boundaries with one another and with the inner and outer surface meshes from the nozzle. All of these meshes contain 52 nodes in the circumferentially direction. The 16 meshes of the outer ring have 60 nodes in the radial direction while the 16 meshes of the inner ring have 59 nodes in the radial direction. Traveling in the streamwise direction, the first 24 meshes have 50 nodes in the axial direction and the final 8 meshes have 53 nodes the axial direction. The maximum number of nodes in this group is 165,360 while the minimum number of nodes in this group is 153,400. Figure 5.11 shows a slice of the shear layer resolving meshes to highlight the mesh density used to resolve the separated shear layers. As the flow exits the nozzle, the resolution normal to the shear layer is set to $4 \times 10^{-5}$ and gradually expands until the minimum resolution is approximately $3 \times 10^{-3}$ at 2.7 diameters downstream of the nozzle exit.

![Figure 5.10: Meshes which make up the surface of the nozzle lip and the separated shear layer. (a) exploded view of all 32 meshes with all surfaces visualized. (b) assembled view with all 32 surfaces visualized. The surface of the nozzle geometry is also visualized in the background for reference.](image)

All of the previous meshes share block-to-block boundaries and are constructed in a radial pattern about the central axis of the nozzle. Since the blocks must have a structured nature, they cannot converge to a point to resolve the flow at the central core of the nozzle. This undesirable geometry is avoided by resolving the central core of the flow with rectangular structured meshes, shown in figure 5.12.
Figure 5.11: Side view of the nozzle surface meshes and a slice through the central axis of the meshes which resolve the shear layer exiting the jet. (a) full view of the meshes which resolve the shear layer. (b) magnified view of the shear layers exiting the nozzle.

The resolution of the meshes in the direction of the nozzle axis is matched to the resolution of the surrounding surface-resolving meshes. Each of these meshes have 44 nodes in directions normal to the jet axis. The upstream mesh (green) has 80 nodes in the axial direction, while the nozzle exit mesh (blue) and the wake mesh (red) have 75 nodes and 83 nodes respectively in the axial direction. This gives the upstream mesh 154,880 total node points, the nozzle exit mesh 145,200 node points, and the wake mesh 160,688 node points.

Downstream of the nozzle exit, six refinement meshes have been placed to resolve the separated jet wake. These blocks can be seen in figure 5.13 along with the nozzle surface meshes for reference. The first four blocks share block-to-block boundaries with one another and overlap with the fifth refinement mesh (green). The dimensions of the first four meshes are 86 node points in the nozzle axis direction and 43 points in each of the lateral directions for a total of 159,014 node points in each mesh. The node points are spaced to provide a uniform dimensionless cell size of $4.16 \times 10^{-2}$. The fifth refinement mesh expands the individual cells to a dimensionless size of $6.8 \times 10^{-2}$. This allows the fifth mesh to occupy an equivalent volume as the previous four meshes with 53 node points in each direction for a total
Figure 5.12: View of the three rectangular meshes used to resolve the flow in the center of the nozzle. (a) assembled view of the meshes and a wire frame representation of the nozzle and shear layer-resolving meshes. (b) magnified view of the central core meshes near the nozzle exit. These meshes are refined in the direction of the nozzle axis to match the level of resolution in the nozzle geometry-resolving meshes.

of 148,877 nodes. The fifth mesh also overlaps with the sixth mesh, which has the same number of node points as the fifth mesh, but occupies a larger volume due to an increased dimensionless cell size of $9.05 \times 10^{-2}$. The side view emphasizes that the volume of space occupied by the first 5 meshes occupies $1 < \tilde{x} < 7.6$ and $-1.77 < \tilde{y}, \tilde{z} < 1.77$. The sixth mesh extends the refined volume to $7.3 < \tilde{x} < 12$ and $-2.35 < \tilde{y}, \tilde{z} < 2.35$ which allows the jet wake to be resolved at least 11 diameters downstream of the nozzle exit.

Finally, the region of space between the refinement meshes and the cylindrical far-field mesh is resolved with 10 cylindrical refinement meshes, shown in figure 5.14. The first 8 quarter-round meshes share block-to-block boundaries with one another and with the nozzle and shear-layer resolving meshes of figures 5.8 and 5.10. As such, each of these meshes has 52 node points in the circumferential direction and they all share 17 node points in the radial direction. The four upstream meshes (larger quarter-round meshes) have 179 node points along the nozzle axis for a total of 158,236 node points each. The four downstream meshes
Figure 5.13: View of the 6 rectangular meshes used to resolve the jet wake. (a) assembled view of the 6 wake-resolving meshes along with the nozzle surface meshes in the background for reference. (b) side view of the 6 wake-resolving meshes showing the relationship to the nozzle and the shear-layer resolving meshes.

(smaller quarter-round meshes) have 180 node points along the nozzle axis which gives a total of 159,120 node points each. The two half-round outer-wake resolving meshes (blue and red) each have 111 nodes in the nozzle axis direction, 23 nodes in the radial direction, and 61 nodes circumferentially for a total of 155,733 nodes. A side view of these meshes, along with the nozzle surface meshes for reference, are shown in figure 5.15. This figure also provides wire-frame representations of the shear layer and wake-resolving meshes to show how these cylindrical refinement meshes overlap these domains. Since the first 8 meshes (represented by the red and blue slice planes) share block-to-block boundaries with the outer nozzle meshes, the refinements required to resolve the nozzle lip are continued outward in these meshes.

The resulting overset system is passed through PEGASUS to calculate the optimized level-2 interpolation stencils which CFDSHIP will use to obtain an optimized flow solution. A slice through the optimized domain, showing only the valid field points upon which the flow solution is calculated and only a single fringe of points which are used to pass information from one mesh to the next, are shown in figure 5.16. While this figure only shows a single fringe for clarity, the PEGASUS
Figure 5.14: View of the 10 cylindrical meshes used to resolve the flow outside of the nozzle external boundary layers and the outer regions of the jet wake.

Figure 5.15: Slice of the upper half of the cylindrical meshes which resolve the flow outside of the nozzle external boundary layer and the outer regions of the jet wake. The nozzle surface meshes are shown for reference along with wire-frame representations of the shear-layer and wake-resolving meshes.

software was used to return a double fringe for all field points which is required by CFDSHIP. Figure 5.16 shows how the cylindrical far-field mesh overlaps with the far-field core mesh and the local refinement meshes.

Due to their density, the refinement meshes appear as solid black regions when
Figure 5.16: Slice through the central axis of all overset domains after PEGASUS overlap optimization.

viewing the entire system. To get a better look at the detail of these blocks, it is necessary to magnify them as in figure 5.17. Even within this figure, the densities of the meshes within and around the nozzle exit cannot be resolved from this view. One of the problems with using block-to-block meshes to resolve the nozzle geometry is that the number of mesh points circumferentially and axially must be the same for each block so that mesh points are coincident at the block-to-block boundaries. This requires that the high mesh density needed to resolve the nozzle lip surface is continued in the radial direction, increasing mesh density in regions far from the nozzle where little unsteadiness is expected within the flow. The result is an overly expensive computational domain and large discrepancies between the cell size of the cylindrical refinement meshes and the cylindrical far-field mesh. This large discrepancy in mesh spacing can cause numerical issues or unphysical flow behavior if unsteadiness is present. Therefore the interface must be taken as far from the unsteady portion of the flow as is practical. Unfortunately, this substantially increases the cost of the computation. Results of computations with this computational domain indicate that the distance of this overlap is sufficiently far from the region of interest for quiescent jet flows.
Examination of the mesh structure in the shear layer refinement region requires considerably more magnification. A magnified view of this region is shown in figure 5.18. The solid white region is the upper portion of the nozzle lip. The boundary layer resolution of the inside, outside, and nozzle lip are visible, along with the mesh clustering to resolve the internal and external separated shear layers. The shear layer mesh clustering expands slowly with downstream distance from the nozzle exit to minimize the jump in cell size where the shear layer meshes overlap with the wake refinement meshes (visible in figure 5.17).

Before this computational domain can be used by CFDSHIP to compute the jet flow, boundary conditions must be proscribed to all boundary surfaces of the meshes. The mathematical definition of the boundary conditions is described in section 5.1 and the placement of these boundary conditions is shown in figure 5.19. No-slip boundary conditions (shown in green) are applied to all of the surfaces which make up the nozzle as well as the upstream face of the cylindrical far-field mesh. The downstream face of the cylindrical far-field mesh is associated with the exit boundary condition and the outer radial surface of this mesh is associated with the far-field boundary condition. The upstream face of the meshes which fill
the nozzle are associated with the inlet boundary condition to provide a source of momentum to the domain.

5.4 Performance

CFDSHIP uses coarse-grain parallelism, where each mesh is solved on its own processor, to distribute the work load of computations over multiple processors. Since the present computational domain is comprised of 69 meshes, 69 processors are required to obtain a solution. The solutions which will be presented in the following sections were computed on a United States Army Research Laboratory supercomputer which has 2048 3.6 GHz Intel Xeon EM64T processors with 2GB of memory per processor. This hardware can compute 2000 time steps worth of data in 20 hours of wall time, or 1,380 CPU hours. The solutions that will be shown are the product of about 70,000 time steps. Thus the total expense for the CFD results of the jet flow is 48,300 CPU hours, or 700 hours of computation times 69 processors. Computation of the solution also required a total of 25.2 gigabytes of memory, which averages out to a little over 365 megabytes per processor. These
The bubble dynamics code faces a different set of limitations for speed and scalability. The code must first read the fluid dynamics solution from disk before the bubble dynamics calculations can begin. Only one processor reads the data, leaving any additional processors idle. Once the solution is loaded into memory, the available processors evenly distribute the workload to compute the bubble dynamics for that time step. Once the bubble dynamics computations are completed, the code repeats the process and begins to read the next time step from disk. The time required to read the solution from disk will not be reduced unless faster hardware is used. Figure 5.20 shows that the total time required to read the data from disk and then broadcast the data to all of the processors actually increases with the number of processors used. While the time required to read from disk is increasing as processors are added, the amount of time associated with the bubble dynamics calculations decreases. This occurs because the number of bubbles being computed...
by each processor reduces as the total number of processors increases. Figure 5.21 shows that the actual amount of time required to perform the bubble dynamics calculations is directly proportional to the number of bubbles being handled by any given processor. While increasing the number of processors used to obtain a solution will generally decrease the computing time, there is a limitation to how much the computation time can be reduced due to the large amount of disk access and message passing required for this problem.

Figure 5.20: Time required to read the fluid dynamics solution from disk and broadcast to available processors with MPI versus the number of available processors.

5.5 Bubble Dynamics Code Verification

The Lagrangian bubble dynamics code written for this project requires verification testing to ensure that it is returning accurate solutions to the governing equations. Verification of the bubble dynamics code will consist of comparisons with known solutions for the governing equations. The two main functions of the bubble dynamics code are to numerically solve the EoM for the bubble trajectory and the R-P equation for the bubble size. Both of these equations are solved using the
Figure 5.21: Time required to compute the bubble dynamics versus the number of bubbles being computed per processor.

4th-order Runge-Kutta (RK4) method of integration. The RK4 solution of the EoM will be compared against an analytical solution to the terminal velocity of a bubble. The R-P equation has no general solution and will require a different approach. Solutions of the R-P equation from the bubble dynamics code will be compared against solutions obtained from Mathematica.

5.5.1 Equation of Motion

The motion of a rising bubble in an infinite liquid is driven by a buoyancy force and resisted by the liquid inertia, surface tension, and fluid viscosity[83]. Research has shown that nuclei bubbles behave as solid spheres, and therefore do not experience a resistive force from surface tension. If we assume that there are no hydrodynamic pressure gradients in the infinite liquid, then the terminal velocity of the bubble can be found by setting the acceleration of the bubble to zero and solving for the velocity. The dimensional EoM from chapter 4 is equation 4.84. Setting the acceleration to zero and neglecting bulk liquid velocity, pressure gradients, lift
forces, and volume variation forces gives,

\[ 2g_i = -\frac{3}{4} \frac{C_D}{a} V_i^2 \]  \hspace{1cm} (5.20)

Equation 5.20 shows that at equilibrium, the drag and buoyancy forces must be equal and opposite. This equation can be rearranged to solve for the magnitude of the terminal velocity, \( V_t \), where,

\[ V_t = \sqrt{\frac{8}{3} |\vec{g}| \frac{a}{C_D}} \]  \hspace{1cm} (5.21)

To solve for the velocity, the nonlinear drag coefficient must be known. The bubble dynamics code uses an empirical formulation for \( C_D \) given by White[75] and listed in equation 4.85. Unfortunately, \( C_D = f(V) \), making the problem nonlinear. However, the terminal velocity can be found in an iterative fashion. The process involves using an initial guess to calculate \( C_D \). This value is substituted into equation 5.21, where \( V_t \) is solved. The new value of \( V_t \) is then used to calculate and updated \( C_D \) and the process is repeated until the process converges to an unchanging value of \( V_t \). This iterative approach was found to give the terminal velocity as long as a suitable initial guess was given.

The terminal velocity predicted by equation 5.21 was calculated for a 10\( \mu m \), 100\( \mu m \), and a 1\( mm \) bubble. Numerical simulations for these bubble sizes were also performed with the Lagrangian bubble dynamics code. The bubbles were released from rest in a still fluid and allowed to move according to the full EoM. The results of the numerical simulations are shown in figure 5.5.1 along with the terminal velocities predicted by equation 5.21 (labeled at “theoretical”). The figure shows that the bubble dynamics code predicts that all of the bubbles reach their terminal velocities after an initial acceleration and deceleration. These results confirm that the RK4 algorithm for the EoM is implemented correctly within the bubble dynamics code.

### 5.5.2 Rayleigh-Plesset Equation

The R-P equation is a nonlinear second-order ordinary differential equation (ODE) with no general solutions. The RK4 algorithm in the bubble dynamics code was
Figure 5.22: Terminal velocity calculations for bubbles rising due to buoyancy. The computed data is from the bubble dynamics code and the “theoretical” data refers to the solution given by equation 5.21 validated through comparisons with solutions obtained by Mathematica. Mathematica is a commercially available general purpose mathematics code with the ability to solve ODEs. Solutions were obtained for single bubbles exposed to time-varying pressure fields. Figure 5.23 shows a sample of the agreement which was
typical for the bubble dynamics solutions and the Mathematica solutions. The
sample presented in figure 5.23 represents a 50µm bubble released into a time
varying pressure field with a frequency of 4000Hz. The two solutions are nearly
identical with their plotted lines covering one another on the graph. A large num-
ber of sample cases were compared to ensure that the RK4 algorithm was properly
implemented in the bubble dynamics code. In all cases, agreement like that shown
in figure 5.23 was obtained.

Figure 5.23: Comparison of solutions of a single bubble driven by a time vary-
ing pressure field. The red line represents the solution obtained from the bubble
dynamics code while the black line represents the solution obtained from Mathe-
matica.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>release_box</td>
<td>Character</td>
<td>“box”</td>
<td>Defines the shape of the release box (Options: “box” or “wedge”)</td>
</tr>
<tr>
<td>inlet_xmin</td>
<td>Double</td>
<td>-1.0</td>
<td>Release box face normal to the x-axis and located at the minimum x-value</td>
</tr>
<tr>
<td>inlet_xmax</td>
<td>Double</td>
<td>0.0</td>
<td>Release box face normal to the x-axis and located at the maximum x-value</td>
</tr>
<tr>
<td>inlet_ymin</td>
<td>Double</td>
<td>-1.0</td>
<td>Release box face normal to the y-axis and located at the minimum y-value</td>
</tr>
<tr>
<td>inlet_ymax</td>
<td>Double</td>
<td>0.0</td>
<td>Release box face normal to the y-axis and located at the maximum y-value</td>
</tr>
<tr>
<td>inlet_zmin</td>
<td>Double</td>
<td>-1.0</td>
<td>Release box face normal to the z-axis and located at the minimum z-value</td>
</tr>
<tr>
<td>inlet_zmax</td>
<td>Double</td>
<td>0.0</td>
<td>Release box face normal to the z-axis and located at the maximum z-value</td>
</tr>
<tr>
<td>inlet_ri</td>
<td>Double</td>
<td>0.0</td>
<td>Inner radius of release box “wedge” (see figure 5.5)</td>
</tr>
<tr>
<td>inlet_ro</td>
<td>Double</td>
<td>1.0</td>
<td>Outer radius of release box “wedge” (see figure 5.5)</td>
</tr>
<tr>
<td>theta_start</td>
<td>Double</td>
<td>0.0</td>
<td>Starting angle (in degrees) of release box “wedge” (see figure 5.5)</td>
</tr>
<tr>
<td>theta_end</td>
<td>Double</td>
<td>360.0</td>
<td>Ending angle (in degrees) of release box “wedge” (see figure 5.5)</td>
</tr>
<tr>
<td>yaxis_offset</td>
<td>Double</td>
<td>0.0</td>
<td>Offset from the y-axis for the central axis of release box “wedge”</td>
</tr>
<tr>
<td>zaxis_offset</td>
<td>Double</td>
<td>0.0</td>
<td>Offset from the z-axis for the central axis of release box “wedge”</td>
</tr>
</tbody>
</table>

Table 5.10: Parameters which can be set to define an inlet box in which the nuclei bubbles are initially randomly distributed.
Chapter 6

Circular Jet Simulations

The main goal of this thesis is to understand cavitation scale effects in high-Reynolds number circular jet flows. To understand cavitation scale effects in jet flows, it is first necessary to understand the dynamics of the jet flow. In this chapter, I will be presenting the results of a URANS simulation, a DES, and an ILES of a circular jet operating at \( Re = 500,000 \). Figure 6.1 shows the vorticity of the fluid on a slice through the central axis of the jet for each of these methods. It is apparent from this figure that each method gives different results. The goal of this chapter is to evaluate the results from these three different models and determine which result is the closest to experimental data. Once the best model is determined, this model will be used as the basis for the cavitation inception studies which will discussed in the next chapter.

6.1 Operating parameters

Before discussing the results of the flow, we must first define the parameters of the problem. The geometry, dimensionless boundary conditions, and dimensionless initial conditions were discussed in chapter 5. These parameters alone are insufficient to fully define the problem. In addition to these parameters, the physical Reynolds number and dimensionless turbulent Reynolds number are needed in equation 4.12. For the present work, a physical Reynolds number of 500,000 is used so that the results obtained from the numerical simulations can be compared against experimental data of the same Reynolds number. This data was obtained
Figure 6.1: Vorticity on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents $x/D_J$ coordinates while the ordinate represents $r/D_J$ coordinates.

from studies at the Garfield Thomas Water Tunnel[16, 12] and studies performed at The Johns Hopkins University[14, 20]. For the URANS and DES results, the
turbulent Reynolds number is determined from the blended \( k - \epsilon / k - \omega \) turbulence model of Menter. This model is defined by equations 4.16 through 4.18. While the URANS model uses the full value of the turbulent eddy viscosity, the DES model uses a reduced value of the turbulent eddy viscosity in regions where the grid is sufficient enough to resolve the larger turbulent fluctuations. The ILES forces \( Re_t \to \infty \) and relies on the implicit dissipation of the numerical schemes to account for the unresolved turbulent dissipation.

The governing equations are used in their nondimensional form. This allows the results to pertain to all geometrically similar flows which operate at a Reynolds number of 500,000. If the kinematic viscosity of seawater is taken to be \( 1.17 \times 10^{-6} \text{m}^2/\text{s} \), the velocity of a water jet operating at \( Re = 500,000 \) becomes a function of the jet’s diameter. As a reference, the physical velocities for several different jet diameters are shown in table 6.1.

<table>
<thead>
<tr>
<th>Jet Diameter</th>
<th>Jet Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0254 meter (1 inch)</td>
<td>23.03 meters per second</td>
</tr>
<tr>
<td>0.1016 meter (4 inch)</td>
<td>5.76 meters per second</td>
</tr>
<tr>
<td>0.5 meter</td>
<td>1.17 meters per second</td>
</tr>
<tr>
<td>1.0 meter</td>
<td>0.59 meters per second</td>
</tr>
</tbody>
</table>

Table 6.1: Table of jet velocities which correspond to jet diameter at a Reynolds number of 500,000 and assuming a standard value of kinematic viscosity for seawater.

It is important to point out that experimentally, it is not always possible to maintain dynamic similarity between a model flow and a prototype flow. As shown in table 6.1, small scale jet velocities are much larger than the large scale velocities. As an example, lets assume that a ship would require a 1 meter nozzle for propulsion. If the jet flow exiting the nozzle was operating at 5\( \text{m/s} \) (10 knots), the corresponding Reynolds number of the jet would be \( Re = 4.3 \times 10^6 \). If a scale model test using a 0.1 meter nozzle is to be performed, the reduced-scale jet must have a velocity of 50.3\( \text{m/s} \). This velocity is quite large and may be difficult to achieve in a laboratory setting.

This difficulty with maintaining Reynolds number similarity has led many experiments to be performed using Froude number similarity or even direct velocity
It is important to point out that in these flows, dynamic similarity will not be maintained between the model and prototype flow. In this situation, the dimensionless parameters of the model flow may not represent the dimensionless parameters of the full scale flow. This is a very important point to recognize. The cavitation number is analogous to the dimensionless coefficient of pressure. If we assume that cavitation occurs when the local pressure reaches the vapor pressure of the liquid (not necessarily true) then the cavitation number is equal to the negative of the minimum coefficient of pressure in the flow. If the model flow and prototype flow have different minimum coefficients of pressure, they will also have a different incipient value of cavitation number. Maintaining dynamic similarity eliminates the possibility that the coefficient of pressure will change with length scale and reduces the number of variables which must be considered in the study of cavitation scale effects.

### 6.2 Uncertainty Assessment

One problem with numerical simulations is that they are only approximations of an actual flow. These approximations of the actual flow can be very accurate, very inaccurate, or more typically somewhere in between. There are many factors which determine how well the numerical approximations will match the physical flow. These factors include, but are not limited to, the governing equations, boundary conditions, turbulence models, numerical methods, and the grid.

Historically, it has been customary to perform a verification and validation (V&V) of the numerical code and grid for a given type of flow. Verification involves assessing the numerical uncertainty of the temporal and spatial truncation errors inherent in the numerical methods. This is achieved by performing grid resolution studies in which the solution from systematically refined grids are directly compared. As the grids are refined, the numerical solution should asymptotically converge toward the actual solution of the governing equations. The grid study is used to show that the grid is sufficiently resolved to capture the physics of the flow. One problem with performing verification of chaotic, unsteady data is that different realizations of the flow from different grids will never be exactly the same. In these cases, statistical representations of the flow must be considered. The cost
of V&V for unsteady simulations is considerably higher than steady-state flows because a large sample of unsteady data is needed to ensure that first and second order statistics have converged.

The costs associated with computing the statistics for the URANS, DES, and ILES results of this study are prohibitively large. The computational results shown in this section, including time to reach quasi-steady state, required more than 75,000 CPU hours of computation. Even with this large amount of computation, the results only represent a small amount of physical flow time. The unsteady solutions represent 10,000 time steps worth of data at a dimensionless time step size of 0.001. The resulting unsteady solutions represent only 10 dimensionless units of time, \( \tau_f \). The dimensionless time unit, \( \tau_f \), corresponds to the amount of time it takes a fluid particle traveling at the jet mean exit velocity \( U_\infty = 1 \) to travel 1 jet diameter in length. The dimensionless solution of \( \tau_f = 10 \) represents a flow where a fluid particle with a velocity of \( U_\infty \) would travel 10 jet diameters in distance. Figure 6.2 shows the axial velocity predicted by the DES method along the jet centerline at \( x/D_J = 4 \). This sample captures only 3 cycles of the velocity fluctuation. With such a small sample of the cyclic velocity, calculations of mean and standard deviation values will not be converged. Figure 6.3 shows how the mean value of the velocity at \( r/D_J = 0.5 \) and \( x/D_J = 4 \) changes with sample size. At 2,000 samples (the solution is only sampled every 5 time steps), it is clear that the mean value has not yet converged. Calculation of the standard deviation requires the mean value. Since the mean value has not converged, calculations of the standard deviation are dubious. The lack of convergence in the statistics prohibits a formal V&V. However, the computed mean quantities will provide an approximation of the actual mean quantities and can be used to determine if the CFD solutions is a reasonable approximation of a physical flow.

The second part of V&V is validation. Validation is the process of assessing the modeling uncertainty of the simulation by comparing against experimental data. CFDSHIP is a general purpose CFD code which has been validated against experimental data for a wide array of flows\[79\]. CFDSHIP has already been used to simulate a circular jet flow with the current Reynolds number and geometry. This simulation is documented in a report by Meyer et al\[16\]. However, this earlier work does not represent a validation study, since a validation study requires an
Figure 6.2: Axial velocity at $x/D_J = 4$ for the DES. The solid line represents the velocity on the jet centerline and the dashed line represents the velocity at $r/D_J = 0.5$.

Figure 6.3: Mean value of the axial velocity at $r/D_J = 0.5$ and $x/D_J = 4$ versus sample size. The mean is computed from $\bar{U}_{x,n} = (\sum_{i=1}^{n} U_{x,i})/n$, where $i$ denotes the individual samples, $n$ is the total sample size listed on the abscissa and $U_x$ is the value of the dimensionless axial velocity. There are 200 samples per unit of dimensionless time.

experimental data set with known uncertainty. At the present time, a data set with uncertainty is not available for a circular jet flow at the present Reynolds number. Thus it is not possible to conduct a formal validation study of CFDSHIP for high Reynolds number circular jet flows.

Since it is not possible to perform a formal V&V in the way proposed by Stern
et al. [84] or Oberkampf and Trucano [85], the operating parameters and grid must be specified using best practices obtained from previous experience. Temporal errors are minimized by choosing a time step size which is an order of magnitude smaller than typical time step sizes used for other DES’s performed with CFDSHIP [86, 67]. This time step size was also chosen to minimize the CFL number of the cells at the jet exit. While a CFL number less than 1.0 was not reached in the smallest cells, this time step size maintained a CFL number less than 1 in the regions of the flow that exhibit unsteadiness. The grid was also designed using best practices from other simulations. Sufficient grid points were placed in the near-wall region to ensure that the law-of-the-wall region of the flow is sufficiently resolved.

6.3 Comparison with Literature

While URANS methods have historically been the dominant means of simulating unsteady flows, DES methods have been gaining popularity in the past few years. In this section, the results of the URANS simulations and DES will be compared against jet data reported in the literature. This section will also examine and compare ILES results with the available experimental data. The ILES method was reported by Shur et al. [64] to do a better job than DES methods of simulating the acoustic emissions of high Mach number compressible jet flows. In this section I will show the result of an ILES for the incompressible high-Reynolds number jet flow. The ILES results will be compared with the URANS and DES results to determine which of these methods does the best job of reproducing the jet dynamics.

6.3.1 Statistics

The first measurements to be addressed are the mean axial velocity profiles. Although the sample size from these simulations is too small to calculate the mean axial velocity with confidence, these results provide an indication of what the actual mean velocity profiles would look like. As such, these mean velocity profiles can be compared with the experimental data to determine if the simulations are
providing realistic trends. Hunter Rouse showed that the axial mean velocity profile for a jet is characterized by a mixing layer which is responsible for decelerating the velocity of the essentially irrotational central core and accelerating the surrounding quiescent fluid[4, 6]. The mixing layer increases with distance from the nozzle and eventually spreads to the central core of the nozzle somewhere between $4 < x/D_J < 6$. Until this point, the mean axial velocity along the centerline of the jet is expected to remain constant. After this point, the mean axial velocity of the centerline will decrease proportional to $D_J/x$.

The mean axial velocity profiles computed from the URANS, DES, and ILES results are shown in figure 6.4. These plots correspond to the velocity solutions shown in figure 6.3. The data sampled from these unsteady solutions was mirrored about the central axis to effectively double the amount of data. While this may not improve accuracy much, it does have the benefit of smoothing the data curves. The smoothed mean velocities are plotted at several downstream locations and are overlaid with experimental data from H. Rouse[6]. It is immediately apparent that the URANS simulation does not capture the expected decay of the axial velocity beyond $x/D_J \sim 6$. It is also apparent that the URANS simulation is under predicting the diffusion of the jet shear layer. This under prediction of the shear layer diffusion is especially apparent in the plots of $x/D_J = 8$ and $x/D_J = 10$. Figure 6.4 indicates that the URANS simulation is not returning a physically meaningful solution of the jet flow problem. Because of this, it will not be considered in any further discussions.

While the URANS solution does not give the same trends as the experimental data, the DES and ILES results match the experimental data reasonably well. The DES does show the velocity along the jet axis decaying prior to $x/D_J = 4$. Since the statistics have not fully converged, this initial decay may be biased by long wavelength fluctuations which do not have enough samples to provide convergence with the actual mean value. Keeping this in mind, it is not surprising that we don’t exactly match the trends, but it is encouraging that both methods provide physically realistic velocity profiles. According to these mean velocity profiles, both the DES and ILES results seem to follow the trends observed in experiments.

Plots of standard deviations are shown in figure 6.6 with comparisons to the
Figure 6.4: Comparison of mean velocity profiles obtained from the DES, ILES, and RANS results at various $x/D_J$ locations. Data points are from a 1 inch air jet at $Re = 100,000$.\[6\]
Figure 6.4 (continued)
Figure 6.5: Velocity contours on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents $x/D_J$ coordinates while the ordinate represents $r/D_J$ coordinates.

Experimental data of Rouse\cite{Rouse}. The standard deviations are given by,

$$S_{U_x} = \sqrt{\frac{\sum_{i=1}^{n} U_{x,i}^2 - n\bar{U}_x^2}{n-1}},$$  \hspace{1cm} (6.1)
where \( n \) is the sample size \((n = 2000)\) and \( \bar{U}_x \) is the mean value of the axial velocity. Since the mean values have not converged, any error in the mean is exacerbated by taking the square of the computed values. The computed values of \( S_{U_x} \) match the experimental data well at \( x/D_J = 1 \) and \( x/D_J = 10 \). At \( x/D_J = 2 \) and \( x/D_J = 3 \), the ILES solution shows good agreement with the experimental data while the DES result is significantly higher than the experimental data. At \( x/D_J = 6 \) and \( x/D_J = 8 \), neither solution matches the experimental data well. It is difficult to determine if these results are representative of the actual standard deviation of the computed solutions or if they are significantly biased by the relatively small sample size. Without more computational data, it is not possible to draw conclusions from these standard deviation results. Unfortunately, the computational resources needed to obtain a larger sample are no longer available.

### 6.3.2 Unsteady Dynamics

The mean axial velocity profiles were used to show that the URANS simulation gave physically unrealistic results for the jet flow. These same mean velocity profiles indicate that the DES and ILES results are quite similar. Qualitatively, however, the instantaneous velocity contours for the DES and ILES results are quite different. This section will examine the differences in the unsteady DES and ILES results. The dominant frequency of the velocity and pressure fluctuations are obtained with fast Fourier transforms (FFT) and compared with published values for jets.

#### 6.3.2.1 Axial Velocity

To gain insight into the unsteady dynamics of the jet flow, probes were placed along the central axis and along the radial location \( r/D_J = 0.5 \). The probes were placed at downstream locations corresponding to \( x/D_J = 1, 2, 3, \ldots, 10 \), as shown in figure 6.7. The axial velocity versus time measured at \( r/D_J = 0.5 \) is shown in figure 6.8. It appears that in the range \( 1 \leq x/D_J \leq 4 \), the ILES contains higher frequencies than the DES data. This is expected since the ILES method does not include a subgrid eddy viscosity model which helps to dissipate high frequency turbulent kinetic energy. Plots of the subgrid turbulent kinetic energy are shown in
Figure 6.6: Comparison of computed standard deviation ($S_{Ux}$) for the axial velocity ($U_x$) of the DES and ILES results. Experimental data points are from a 1 inch air jet at $Re = 100,000$. [6]
This figure not only shows that ILES predicts no subgrid eddy viscosity, but also shows that the DES model has a reduced eddy viscosity close to the nozzle because of grid refinement. Figure 6.8 shows that in the range $1 \leq x/D_J \leq 4$ the DES is predicting larger magnitude velocity fluctuations than the ILES. Differences in the two solutions are more evident in plots of the axial velocity along the jet centerline, shown in figure 6.9. These plots show that from $2 \leq x/D_J \leq 5$, the DES results predict a much stronger pulsing along the jet centerline than the ILES results. This is also confirmed in the velocity contours of figure 6.5. While the mean velocity along the centerline is 1 for both solutions, the DES indicates much larger fluctuations about the mean value. These fluctuations are consistent with strong coherent vortex structures which are described in the review article by List. Although much weaker and at a higher frequency, the ILES also shows pulsing along the central axis. The question which must be answered now is: which of these two solutions is a better match for the experimental data. This can best be answered by examining the FFTs of this data and comparing against the published range of Strouhal numbers.

A FFT of the time-varying data will reveal the dominant frequencies which make up the time-varying signal. This dominant frequency is then used to determine the characteristic Strouhal number based upon jet diameter ($St_D = fD_J/U_J$).
Figure 6.8: Velocity versus time for the DES and ILES results at $r/D_J = 0.5$. 
Figure 6.8 (continued)
Figure 6.9: Velocity versus time for the DES and ILES results along the jet axis ($r/D_J = 0$).
Figure 6.9 (continued)
Figure 6.10: Turbulent kinetic energy (TKE) contours on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents $x/D_J$ coordinates while the ordinate represents $r/D_J$ coordinates. There is no TKE in the ILES solution because no subgrid turbulence model is used.
However, since the data is already nondimensionalized, the FFT of the dimensionless axial velocity will return the dimensionless frequency, or $St_D$ for the signal. Thus no conversions are required to obtain the characteristic Strouhal number for the jet flow. While the Strouhal number has been shown to be sensitive to the initial shear layer thickness\cite{21}, published results indicate that $St_D$ will be in the range $0.3 \leq St_D \leq 0.5$\cite{54} for high-Reynolds number incompressible jets. FFTs for the DES and ILES axial velocities are shown in figures 6.11 and 6.12 for the locations at $r/D_J = 0.5$ and $r/D_J = 0$, respectively. These FFTs were calculated using GNU Octave (\url{www.gnu.org/software/octave}), which uses the FFTW library from \url{www.fftw.org}\cite{87}. An example of the Octave commands which were used to generate the following FFT plots is listed in appendix A. The resulting FFTs for $r/D_J = 0.5$ show that the ILES and DES results are resolving similar frequencies at this radial location. At $x/D_J = 1$, the ILES shows more energy at higher frequencies than the DES, an outcome that was expected from the direct observation of the time-varying velocity signal at this location. The DES signal has a peak value at $St_D = 0.6$ while the ILES has peaks at $St_D = 0.2$ and $St_D \sim 1$. Further downstream, the DES shows clear peaks in the Strouhal number at $St_D = 0.3$ up until $x/D_J = 6$, while the ILES consistently shows a peak at $St_D = 0.2$ beyond $x/D_J = 3$. From $1 \leq x/D_J \leq 3$, the ILES shows peaks between $0.5 \leq St_D < 2$.

As with the direct observations, the FFTs of the velocity along the jet axis (figure 6.12) show a larger discrepancy between the two solutions. The FFTs at $x/D_J = 1$ show that the majority of the signal for the DES is associated with $0.4 \leq St_D \leq 0.6$ whereas the ILES has a peak at $St_D = 0.3$. From $2 \leq x/D_J \leq 8$ the DES shows peak energy associated with $St_D = 0.4$ and $St_D = 0.3$, whereas the ILES shows peaks at $0.2 \leq St_D \leq 0.6$. We should also note that until $x/D_J = 5$, the energy associated with the peak Strouhal number is significantly lower for the ILES results. This indicates that the ILES fluctuations are smaller. Since both methods result in peak frequencies within the expected range, it is not clear from this data which of the two solutions is better.

While the mean velocity profiles and FFT data indicate that the DES and ILES results are quite similar, the unsteady velocity contours of figure 6.5 and the unsteady vorticity contours of figure 6.1 indicate otherwise. The DES gives a
Figure 6.11: FFT of the velocity signal for the DES and ILES results at \( r/D_J = 0.5 \).
Figure 6.11 (continued)
Figure 6.12: FFT of the velocity signal for the DES and ILES results along the jet axis ($r/D_J = 0$).
Figure 6.12 (continued)
thin shear layer which succumbs to a Kelvin-Helmholtz instability around \( x/D_J = 0.25 \) and rolls up into coherent vortex rings. The ILES gives a thin shear layer which becomes unstable almost immediately \( (x/D_J \sim 0.05) \) upon exiting the jet and rolls up into vortex rings. While the DES vortex rings remain coherent for many diameters downstream, the ILES vortex rings quickly break down into 3-dimensional flow. The coherence of the ILES vortex rings is not obvious beyond \( x/D_J = 1 \). The review of jet dynamics in Chapter-1 indicates that jet flows are very sensitive to initial shear layer thickness. The differences in the initial shear layers from the URANS, DES, and ILES results are shown by plotting velocity profiles at the jet exit. Figure 6.13 shows that the URANS and DES solutions are very similar, which is expected. These two methods use the same sub-grid turbulence model. The minor differences between the URANS and DES results are due to sufficient grid density upstream of the exit which leads to a small reduction of the turbulent eddy viscosity in the DES model. The ILES velocity profile is quite different from the URANS and DES profiles. It approaches the centerline velocity much faster than the other solutions. Cerutti et al.\cite{21} have shown that the spacing of vortex rings is dependent upon the initial thickness of the jet shear layer. These authors found that thinner shear layers will reduce the \( x/D_J \) location where vortex rings form. The simulation performed by the authors was axisymmetric and therefore the circumferential instabilities which lead to vortex ring breakup was not witnessed. Cerutti et al. also found that Strouhal number based upon the shear layer thickness was rather constant with a value of \( St_\delta = 0.1 \). This Strouhal number is defined as,

\[
St_\delta = \frac{f\delta}{U_J} = St_D \frac{\delta}{D_J}, \quad (6.2)
\]

where \( \delta \) is the shear layer thickness. From figure 6.13, \( \delta_{DES} \sim 0.07 \) and \( \delta_{ILES} \sim 0.015 \). The Strouhal numbers based on diameter at \( x/D_J = 1 \) are found from figure 6.11 to be 1.4 for the DES results and a strong peak occurs at 6.0 for the ILES results. Using these values, \( St_{\delta,DES} = 0.098 \) and \( St_{\delta,ILES} = 0.09 \). These results are consistent with those of Cerutti et al.
Figure 6.13: Velocity profiles at $x/D_J = 0.01$ for the URANS, DES, and ILES results.

6.3.2.2 Pressure

For the problem of cavitation inception, it is more critical that the simulations capture the pressure fluctuations which occur in the physical flow. The time history of the coefficient of pressure from the DES and ILES results are shown in figure 6.14. Instantaneous snapshots of the dimensionless pressure fields ($C_P/2$) for the URANS, DES, and ILES results are shown in figure 6.15. Measurements of the time histories were taken at $r/D_J = 0.5$. The time histories show that the ILES has much more high frequency content from $1 \leq x/D_J \leq 4$ and the DES shows a strong low frequency content which indicates strong coherent structures up through $x/D_J = 6$. The DES pressure also shows much larger amplitude than the ILES. The low frequency, large amplitude pressure fluctuations correspond to vortex rings which can clearly be seen at $x/D_J = 4.5$ in figure 6.3.2.2. The ILES solution in figure 6.3.2.2 shows little indication of coherent vortex rings beyond
The frequency content of these pressure measurements is obtained from the FFTs shown in figure 6.16. At $x/D_J = 1$, there are two dominant frequencies for the DES, $St_D = 0.6$ and $St_D \sim 1.5$. The ILES shows a peak at $St_D \sim 1.5$ and a spike at $St_D = 6$. From $2 \leq x/D_J \leq 6$ the DES shows a peak frequency between $0.3 \leq St_D \leq 0.4$, which in right in line with experimental data. The ILES has a peak between $0.2 \leq St_D \leq 0.6$, which is the same result as the velocity measurements.

The measurements of axial velocity and pressure fluctuations indicate that both solutions resolve the dominant frequencies which were observed in experimental studies. However, the DES results in more energy within the dominant frequencies than the ILES. Because of this difference in energy, the magnitude of the pressure fluctuations may be considerably different between the two solutions. Ran and Katz[14] measured the minimum coefficient of pressure in a jet of $Re = 500,000$. These authors found the minimum coefficient of pressure to be $C_{P,min} = -0.97$. This value was recorded at $x/D_J \sim 2.5$. A plot of $C_P$ from 100 probes evenly spaced in the DES between $0 \leq x/D_J \leq 10$ at $r/D_J = 0.5$ is shown in figure 6.17. This figure shows that the minimum $C_P$ is about $-0.81$. This occurs at $x/D_J \sim 4$, which is further downstream than the minimum value recorded by Ran and Katz. A search of the entire DES data set reveals that the minimum computed value of $C_P$ is $-1.349$ and is found at $x/D_J \sim 4.5$. The experimentally measured $C_{P,min}$ falls between the minimum value from figure 6.17 and the minimum value from the full domain search. Coefficient of pressure values in the region of $x/D_J = 2.5$ were found to be close to those observed by Ran and Katz.

The $C_P$ measurements which were taken from the ILES are shown in figure 6.18. In this simulation, the minimum pressure is on the order of $-0.72$ and occurs around $x/D_J = 0.3$. Filtering out the pressure signals from $0 < x/D_J < 1$ (figure 6.19) shows that the minimum measured pressure in the remainder of the flow is on the order of $-0.42$. This is about $1/2$ of the measured value of $C_{P,min}$ in the downstream flow of the DES and less than $1/2$ the value measured in experiments. A full search of the ILES data set reveals that the minimum computed $C_P$ value is $C_{P,min} = -1.19$ and occurs at $x/D_J = 0.28$, $r/D_J = 0.5$ (but at a different circumferential location than the probes). While the value of $C_{P,min}$ is also of the
Figure 6.14: Coefficient of pressure ($C_P$) versus time for the DES and ILES results at $r/D_J = 0.5$. 
Figure 6.14 (continued)
Figure 6.15: Dimensionless pressure (1/2 coefficient of pressure) on a slice through the central axis of the jet for the (a) URANS simulation, (b) the DES, and (c) the ILES. The abscissa represents $x/D_J$ coordinates while the ordinate represents $r/D_J$ coordinates.
Figure 6.16: FFT of $C_P$ for the DES and ILES results at $r/D_J = 0.5$. 
Figure 6.16 (continued)
Figure 6.17: Coefficient of pressure data from 100 probes evenly spaced between $0 \leq x/D_J \leq 10$ in the DES results.
same order of magnitude as the experimental measurements, the location of the measurement is quite different.

Figure 6.18: Coefficient of pressure data from 100 probes evenly spaced between $0 \leq x/D_J \leq 10$ in the ILES results.

While $C_{P,\text{min}}$ from the DES is due to vortex ring interactions, the minimum pressure in the ILES occurs in the initial vortex rings which develop from the Kelvin-Helmholtz instability of the shear layer. The initial vortex rings (and therefore $C_{P,\text{min}}$) occur within the first 1/2 diameter from the jet origin. After this point, the vortex rings break up into a three-dimensional flow. There are no obvious vortex ring structures beyond $x/D_J = 1$. The ILES results lack the vortex rings to interact and produce minimum pressures at $2.5 < x/D_J < 4.5$ as observed in experiments. The lack of coherent vortex rings beyond $x/D_J = 1$ and the values of $C_{P,\text{min}}$ indicate that the ILES is not reproducing the unsteady pressure of the jet flow.

The literature review in chapter 2 indicates that a circular jet flow is expected to have coherent vortex rings. Vortex rings are obvious in the DES contours of figure 6. A convenient method of accounting for the dynamics of the vortex rings in.
a jet is suggested by Grinstein et al.\cite{55}. These authors presented a plot with the vortex core axial locations along the abscissa versus time step along the ordinate. The result is a 2-dimensional plot which shows the spatial and temporal dynamics of the vortex rings for their jet simulation. Results from the DES are plotted in the same manner and shown in figure\ref{fig:6.20}. The data reveals that the vortex rings form around \(0.4 < x/D_J < 0.8\), which is consistent with the findings Gopalan et al.\cite{20} and the numerical simulations of Grinstein et al.\cite{55}.

Figure\ref{fig:6.20} also indicates the location of vortex pairing. According to this figure, two vortex pairings typically occur, the first between \(1 \leq x/D_J \leq 2\), and the second between \(2 \leq x/D_J \leq 3.5\). These locations agree well with the experimental data of Ran and Katz\cite{14} which showed the first vortex pairing to occur in the range \(1.0 < x/D_J < 1.85\) and the second vortex pairing to occur in the range \(1.5 < x/D_J < 3.5\). The velocity of the vortex rings is given by the slopes of the vortex trajectories in figure\ref{fig:6.20}. The vortex rings in the DES have a dimensionless velocity on the order of \(V_{VR} = 0.6\) (%60 of the jet velocity) which is very close to
Figure 6.20: Trajectories and pairing of coherent vortex rings which form within the DES.

the value of %65 of the jet velocity by Lau and Fisher[56].

6.4 Summary

This chapter showed the results of three different CFD simulations of a $Re = 5 \times 10^5$ circular jet flow. The three CFD simulations consisted of an unsteady Reynolds-averaged Navier-Stokes (URANS) simulation, a detached-eddy simulation (DES), and an implicit large-eddy simulation (ILES). All three of these numerical simulations were conducted on the same grid to allow direct comparison of the results. Due to the size of the grid, these computations proved to be quite expensive. The
large size also necessitated a large amount of disk space to store the transient solutions. Due to these large requirements, the transient simulations were limited to 10,000 time steps with a dimensionless time step size of 0.001. This resulted in a sample which contains 10 units of dimensionless time (i.e. a fluid element traveling at the jet velocity will traverse 10 jet diameters). This data amounted to a relatively small sample of the jet flow dynamics and resulted in statistical values which were not converged.

Even though the statistics have not converged, the mean quantities are an approximation of the actual mean. Comparisons of the computed mean values indicated that the URANS simulation was not representing the experimental data. The shear layer diffusion is under-predicted and the velocity along the jet axis does not exhibit the proper decay beyond $x/D_J = 6$. Because of its failure to simulate the jet velocity field, the URANS simulation was ruled out as a possibility for use in the cavitation study.

Both the DES and ILES provide reasonable values for the mean velocity profiles and characteristic Strouhal numbers. Based upon these quantities alone, both methods appeared to be acceptable. However, upon examination of the initial shear layers and transient pressure fields, the ILES data did not match expected trends. Experimental data indicates that coherent vortex structures will persist through $x/D_J = 6$. While the DES data clearly indicates the presence of coherent structures through $x/D_J = 6$, the ILES solution only exhibits vortex rings through $x/D_J = 1$. The experimental data of Ran and Katz[14] indicated that it was the interaction of coherent vortex rings which is responsible for the low pressure events around $x/D_J = 2.5$. The ILES flow does not clearly resolve coherent vortex rings and therefore lacks this mechanism which generates the low pressures. While resolved turbulent fluctuations could make coherent vortex rings more difficult to detect visually, the presence of a vortex ring should result in a periodic surge in the velocity along the jet centerline. A surge, if one exists, is substantially weaker in the ILES results as witnessed by figure 6.9. The lowest pressures in the ILES occurred within 0.3 diameters from the jet origin and was not as low as the pressure events observed in either the DES flow or the experimental data. The DES showed minimum pressure levels in line with experimental observations in the region of $x/D_J = 2$. However, the minimum pressure in the DES occurs
further downstream at $x/D_J = 4$ due to continued vortex ring interactions.

It is interesting that while the ILES flow does not obtain the expected results for the pressure, it does show good agreement with the mean velocity field. While experiments and DES indicate that jet diffusion in the zone of established flow is due to the Kelvin-Helmholtz instability and vortex ring interactions, the ILES simulation gives nearly identical jet diffusion results from a different mechanism. For the ILES flow, the shear layer diffusion is primarily due to large 3-dimensional turbulent-like eddies. This should serve as a warning to using ILES methods to simulate turbulent flows. These results are an example of an ILES giving believable mean velocity values while not resolving important large-scale coherent structures which significantly affect the pressure field.

Since the ILES results do not capture the coherent vortex rings and transient pressures, the DES data will be used as the basis for the study of cavitation scale effects. In the next chapter, cavitation nuclei bubbles will be released into the DES jet flow and allowed to respond to the unsteady pressure and velocity field. The response of the nuclei bubbles will be monitored to determine the cavitation number associated with incipient cavitation inception. The response of the nuclei will be examined for several different length scales to determine if there is a natural scaling effect due to the bubble dynamics.
Cavitation Scale Effects

The literature review of chapter 2 introduced the problem that engineers have with predicting prototype cavitation inception from scale model tests. In particular, jet flows show an unexplained increase in \( \sigma_i \) with jet diameter. The mathematical model for cavitation inception, derived in chapter 4, is used in this chapter to explain the origins of experimentally observed cavitation scale effects. The results of this study show that the cavitation inception number is likely to change with length and/or velocity of the flow, regardless of whether or not the flow maintains dynamic similarity. A scaling law for the cavitation inception number is derived and is tested by comparing with numerical simulations of cavitation inception and experimental data. The scaling law and numerical simulations will be used to explain the observed relation between \( \sigma_i \) and \( D_J \) for jet flows.

7.1 Equilibrium Scaling Law

The current theory for cavitation inception states that the incipient cavitation number should remain constant between a scale model and a prototype flow\[7, 8\]. Experimental results, however, often show that the incipient cavitation number changes with length scale\[2, 3, 7, 8, 9\]. While Reynolds number effects are also observed, the relationship between \( \sigma_i \) and Reynolds number for jet flows is much weaker than the relationship between \( \sigma_i \) and the length scale of the flow\[14, 21\]. While scale effects are known to exist, there is no theory which allows for a prediction of \( \sigma_i \) for a prototype, based entirely on measurements of \( \sigma_i \) from a scale model.
test. The following sections will probe the relationship between $\sigma_i$ and $L_\infty$ by considering how $\sigma$ is related to the mathematical model for cavitation inception.

### 7.1.1 Simplifications of the Rayleigh-Plesset equation

The mathematical model for a spherical cavitation nuclei was derived in chapter 4. The result was the Rayleigh-Plesset equation; a nonlinear second-order ODE. Recall that the dimensional form of the R-P equation is,

$$a\ddot{a} + \frac{3}{2} (\dot{a})^2 + 4\nu \frac{\dot{a}}{a} = \frac{p_v - p}{\rho} + \frac{p_0}{\rho} \left(\frac{a_0}{a}\right)^3 - \frac{2S}{\rho a}, \quad (7.1)$$

where $a$ is the bubble radius, $\rho$ is the liquid density, $\nu$ is the liquid kinematic viscosity, and $S$ is the surface tension associated with the liquid/gas interface. The pressures $p$ and $p_v$ are the local pressure in the fluid at the location of the bubble and the vapor pressure of the liquid, respectively. The subscript “0” in equation (7.1) indicates that the variable corresponds to measurements of the initial bubble distribution. Throughout the remainder of this work the polytropic exponent “$n$” will be set to 1 as suggested by Ran and Katz[13].

Equation (7.1) has been arranged with all of the dynamic terms on the left-hand side and all of the static terms on the right-hand side. The dynamic terms are zero when the bubble is in equilibrium with the surrounding pressure and only contribute to the equation when an imbalance in the static terms leads to an acceleration of the bubble wall. The first two dynamic terms on the left-hand side of equation (7.1) arise from the acceleration of the fluid surrounding the bubble. The third term on the left-hand side is a viscous damping term.

The static terms on the right-hand side have no dependence on the motion of the bubble and contribute to the equation at all times. The first of these static terms represents the difference in pressure between the vapor within the bubble and the liquid outside of the bubble. The pressure in the liquid just outside the bubble wall is a function of the location of the bubble and the dynamics of the bulk liquid flow (i.e. $p = f(t, \vec{x}, Re)$). The vapor pressure of the liquid will be treated as a constant in the remainder of this work. However, in situations where temperature changes are significant, the change in $p_v$ with temperature must be taken into account. The second term on the right-hand side represents the pressure of the
gas trapped within the bubble. Since the R-P equation assumes no mass transfer across the bubble interface, this term is only a function of the initial conditions and the current radius of the bubble. The final term on the right-hand side of equation 7.1 is the surface tension corresponding to the gas/liquid interface of the bubble. This is an inward acting force and is inversely proportional to the bubble radius. The surface tension, \( S \), is a function of the materials which make up the liquid/gas interface and also a function of the temperature of the interface. Since physical experiments generally use water for the liquid, as well as maintaining constant temperatures, this value will be treated as a constant. Once again, in situations where the temperature or liquid content changes between experiments, the variations of \( S \) must be taken into account.

The nonlinearity of the full R-P equation complicates its study. In an attempt to simplify the equation without losing the significant physics, numerical solutions of the R-P equation were studied to determine which terms in the R-P equation were most significant to the solution. Numerical solutions were examined for a wide array of conditions which included variations in the mean, amplitude, and frequency of the time-varying external pressure, \( p \), as well as variations in the nuclei initial radius. The initial pressure, \( p_0 \), was set by assuming that the initial radius is stable at 110,000 Pa of pressure and 20°C. The study of the R-P equation revealed that the viscous damping term is always small. It was found that eliminating the viscous damping term from the equation had almost no effect on the numerical solution over a wide array of conditions. Exceptions included very high frequencies pressure fluctuations, very small bubble sizes, and the strong collapse and rebound of a nuclei bubble. Away from these situations, the dynamics of the reduced equation are almost identical to the full R-P equation. The numerical studies also revealed that the remaining dynamic terms on the left-hand side of equation 7.1 can also be neglected for a large array of conditions. Removing these dynamic terms results in the equilibrium form of the R-P equation, which is,

\[
p_0 \left( \frac{a_0}{a} \right)^3 - \frac{2S}{a} = p - p_v
\]

Numerical solutions of the full R-P equation and the equilibrium equation are compared in Figure 7.1. The bubbles in this figure are exposed to a time-varying
The minimum pressure, \( p_{\text{min}} \), is varied from 30,000 Pa \((\sigma = 3.0)\) down to the vapor pressure of 2300 Pa \((\sigma = 1.348)\) and the pressure amplitude, \( p_{\text{amp}} \), is held constant with a magnitude of 17 Pa \((\sigma = 0.001)\). The labels in figure 7.1 intentionally show the cavitation numbers corresponding to the minimum pressure to emphasize that the differences between the two solutions are minimum until the cavitation number approaches \(-C_{P_{\text{min}}}\). These comparisons confirm that the equilibrium form of the equation does an adequate job of predicting the nuclei bubble dynamics up until the point where nonlinearities become important. The rise of nonlinearity is typically associated with larger pressure fluctuations which drop below some critical value and/or large frequencies.

Another way to visualize the differences between the full R-P solution and the equilibrium solution is to plot the maximum normalized difference, or error, between the two solutions over a range of pressures and frequencies. Figure 7.2 shows contours of the normalized error for a wide range of frequencies and pressures. The normalized error is defined as,

\[
\text{Error} = \frac{\text{ABS}(\text{MAX}(a_{R-P}) - \text{MAX}(a_{eq}))}{a_{eq}(t = 0)},
\]

where \( \text{MAX}(a_{R-P}) \) is the maximum bubble size from the R-P solution over 25 oscillations, \( \text{MAX}(a_{eq}) \) is the maximum bubble size from the equation 7.2 over 25 oscillations, and \( a_{eq} \) is the solution to equation 7.2 at the maximum pressure \((p_{\text{min}} + p_{\text{amp}})\). The pressure is once again defined by equation 7.3, but the amplitude of the oscillations is increased to 1118 Pa \((10\% \text{ of } C_{P_{\text{min}} \text{ for the jet flow}})\). The abscissa of figure 7.2 lists the difference between the minimum value of the pressure signal and the vapor pressure to emphasize that the nonlinearities do not appear until very close to the vapor pressure, except at high frequency forcing. The frequency of the pressure oscillation is held within the range \( 1Hz \leq f \leq 4000Hz \). This range is representative of the dominant pressure fluctuations which are expected over a wide range of jet length scales. The primary Strouhal number for jets
Figure 7.1: Comparisons of numerical solutions to the full Rayleigh-Plesset equation and the equilibrium form of the Rayleigh-Plesset equation. The black line represents the equilibrium solution and the red line represents the full R-P solution.
(c) $a_0 = 50\mu m, f = 200$

(d) $a_0 = 50\mu m, f = 2000$

Figure 7.1: (continued)
Figure 7.1: (continued)
Figure 7.2: Error between the full R-P equation and the equilibrium equation driven by a time-varying pressure. The mean value and frequency of the oscillating pressure are both altered to determine when the dynamic terms of the R-P equation become important. The fluctuating pressure has a magnitude of $\pm 1118 Pa$ which is equivalent to 10% of $C_{P_{min}}$ for the DES jet solution of chapter 6.

The intensity of the contours represents the error which is defined by equation 7.4.
Figure 7.2: (continued)

(c) 10μm
is from $0.3 \leq St_D \leq 0.5$. Using the value of 0.5 as the limiting case, a 1cm nozzle with $Re = 500,000$ would have a dominant pressure frequency of $3000Hz$, while a 10cm nozzle would have a dominant pressure frequency of $30Hz$. As the length scale increases, the frequency associated with the dominant pressure fluctuations will decrease rapidly (assuming that $Re$ is held constant). Figure 7.2 shows that the equilibrium solution provides a good approximation to the dynamics of nuclei bubbles in the range of expected pressure fluctuations. However, as the minimum pressure approaches the vapor pressure, the errors between the equilibrium solution and the full R-P solution rapidly increase. Still, both solutions are predicting a rapid increase in bubble size. Although the two solutions differ, they will both predict that the cavitation nuclei surpasses the visual criteria of cavitation inception within a narrow range of $p_c$ values. As an approximation, the equilibrium solution will predict cavitation inception at a cavitation number which is very close to the predictions of the full R-P equation. This allows the equilibrium equation to serve as a model for the full R-P equation to represent the dynamics of nuclei bubbles subjected to a time-varying pressure field. However, the approximations of the cavitation number from the equilibrium equation will become progressively worse as the frequency of time-varying pressure increases and/or the initial size of the nuclei bubbles decreases.

### 7.1.2 Origins of scale effects

The equilibrium form of the R-P equation (eq 7.2) can now be examined to understand the origins of cavitation scale effects. Consider two dynamically similar cavitation experiments conducted in the same fluid, but with different physical size. By forcing the fluids to be the same, $S$, $p_v$, $a_0$, and $p_0$ are the same for both experiments. Equation 7.2 can be rearranged to solve for the pressure outside of the bubble. The result is,

$$p = p_v + p_0 \left( \frac{a_0}{a} \right)^3 - \frac{2S}{a},$$

(7.5)

where the constants are written in red. The only variable quantities left are the bubble radius, $a$, and the pressure outside of the bubble, $p$. We now seek the pressure, $p$, which leads to a visible cavity of radius $a_{vis}$. Assuming that the same
size cavity is visible in both experiments, \( a_{vis} \) will also be a constant. Substituting \( a = a_{vis} \) into equation \( e[7.5] \) gives the critical pressure, \( p_c \), which leads to a visible cavity. The critical pressure is given by,

\[
p = p_v + p_0 \left( \frac{a_0}{a_{vis}} \right)^3 - \frac{2S}{a_{vis}}.
\]

Notice that all terms on the right-hand side are now constant for tests conducted in the same fluid. The result is that the critical pressure, \( p_c \), is a constant for all cavitation inception tests conducted in the same fluid.

It has become common practice to use the dimensionless cavitation number, \( \sigma \), to record the conditions when cavitation is detected. The incipient cavitation number is defined as,

\[
\sigma_i = \frac{p_{\infty,i} - p_v}{\frac{1}{2} \rho U_\infty^2},
\]

which compares the magnitude of the reference pressure when inception occurs, \( p_{\infty,i} \), with the vapor pressure, \( p_v \). It has become common practice to assume that cavitation will occur when the minimum pressure, \( p_{\min} \), reaches the vapor pressure. Under this assumption, the incipient cavitation number will be,

\[
\sigma_i = -C_{P_{\min}},
\]

where \( C_{P_{\min}} \) is given by

\[
C_{P_{\min}} = \frac{p_{\min} - p_{\infty}}{\frac{1}{2} \rho U_\infty^2}.
\]

Equation \( e[7.6] \) however, shows that cavitation occurs at a critical pressure, \( p_c \), which may or may not equal \( p_v \). The relation between the incipient value of \( \sigma \) and \( p_c \) can be found by writing the incipient cavitation in terms of \( p_c \). This is accomplished by adding and subtracting \( p_c/(1/2 \rho U_\infty^2) \) from equation \( e[7.7] \). Performing this manipulation and simplifying gives,

\[
\sigma_i = \frac{p_c - p_v}{\frac{1}{2} \rho U_\infty^2} - \frac{p_{\infty,i} - p_{\infty}}{\frac{1}{2} \rho U_\infty^2}.
\]

Recall from chapter \( e[1] \) that the difference between the reference pressure and the pressure at any other point in the flow is independent from the magnitude of
the pressures. If we assume that incipient cavitation will be observed when the minimum pressure in the flow reaches the critical pressure, then \( p_c - p_{\infty,i} = p_{\text{min}} - p_{\infty} \). This substitution uses the fact that the pressure difference is independent of the pressure magnitude. Substituting \( p_c - p_{\infty,i} = p_{\text{min}} - p_{\infty} \) into equation 7.10 and using equation 7.9 gives,

\[
\sigma_i = \frac{p_c - p_v}{\frac{1}{2} \rho U_{\infty}^2} - C_{P_{\text{min}}}. \tag{7.11}
\]

Equation 7.11 shows that the cavitation number will only satisfy equation 7.8 when \( p_c = p_v \). Whenever cavitation is observed at a pressure which is not the vapor pressure, \( \sigma_i \) cannot equal \( -C_{P_{\text{min}}} \). Furthermore, since the quantities \( p_c - p_v \) and \( C_{P_{\text{min}}} \) will be constant for dynamically similar flows conducted in equivalent liquids, \( \sigma_i \) cannot remain constant as the length scale of the flow is varied. The reason is that \( U_{\infty} \) will be inversely proportional to \( L_{\infty} \) when dynamic similarity is maintained. While the current cavitation theory assumes that \( \sigma_i \) should be constant between a scale model and a prototype, the present analysis of the equilibrium R-P equation shows that two dynamically similar flows at different length scales should have different values of \( \sigma_i \).

### 7.1.3 Equilibrium scaling relation

The results of the equilibrium R-P analysis can be used with equation 7.11 to develop an expected scaling relation between \( \sigma_i \) for a scale model flow (\( \sigma_{i,m} \)) and \( \sigma_i \) for a prototype flow (\( \sigma_{i,p} \)). Writing equation 7.11 for the prototype flow gives,

\[
\sigma_{i,p} = \frac{p_c - p_v}{\frac{1}{2} \rho U_{\infty,p}^2} - C_{P_{\text{min}},p}, \tag{7.12}
\]

and for the model flow,

\[
\sigma_{i,m} = \frac{p_c - p_v}{\frac{1}{2} \rho U_{\infty,m}^2} - C_{P_{\text{min}},m}. \tag{7.13}
\]

In these equations, the subscript “\( p \)” specifies that the values are from the prototype flow and the subscript “\( m \)” specifies that the values are from the model flow. The analysis of the equilibrium R-P equation revealed that the quantity \( (p_c - p_v) \) must remain constant between two flows with the same fluid and definition of cav-
itation inception. The magnitude of \((p_c - p_v)\) can be found using equation (7.13) and the result substituted into equation (7.12). Performing this substitution gives,

\[
\sigma_{i,p} = (\sigma_{i,m} + C_{P_{min,m}}) \left( \frac{U_m}{U_p} \right)^2 - C_{P_{min,p}}.
\] (7.14)

This equation provides the prototype incipient cavitation number as a function of the model scale cavitation number and the values of \(C_{P_{min}}\) for the two flows. The relation shows the \(\sigma_{i,p}\) will not be equal to \(\sigma_{i,m}\) unless the cavitation number of the model is identical to the minimum coefficient of pressure for the flow.

Equation (7.14) can be modified to show trends with changes in Reynolds number. If the length scale between the prototype and model is held constant (along with the fluid) but the velocity allowed to vary, equation (7.14) becomes,

\[
\sigma_{i,Re_2} = (\sigma_{i,Re_1} + C_{P_{min,Re_1}}) \left( \frac{Re_1}{Re_2} \right)^2 - C_{P_{min,Re_2}}.
\] (7.15)

This equation shows how \(\sigma_i\) is expected to change with Reynolds number if \(C_{P_{min}} = f(Re)\) is known. To obtain this equation, let all prototype quantities be replaced by \(Re_2\) quantities and all model quantities be replaced by \(Re_1\) quantities. Equation (7.15) also requires use of the relation, \(Re = U_\infty L_\infty / \nu\).

Ideally, the scale model test will be conducted at dynamically similar conditions, but reduced size. Dynamic similarity forces \(Re_p = Re_m\) and \(C_{P_{min,m}} = C_{P_{min,p}}\). Assuming dynamic similarity, equation (7.14) becomes,

\[
\sigma_{i,p} = (\sigma_{i,m} + C_{P_{min}}) \left( \frac{L_{\infty,p}}{L_{\infty,m}} \right)^2 - C_{P_{min}},
\] (7.16)

which gives the change in \(\sigma_i\) with length scale for two dynamically similar flows in the same fluid. This relation allows a prototype flow to be determined if \(\sigma_{i,m}\) and \(C_{P_{min}}\) are known.

Equations (7.14) through (7.16) explicitly assume that the fluids are equivalent (i.e. they have the same physical properties and nuclei distributions) between the model and prototype flow. Implicitly, however, these equations also assume that a sufficient number density of nuclei is present to ensure that the largest
nuclei bubbles always experience the lowest pressure in the flow. Tests where the change in length scale is relatively small (probably one order of magnitude or less) ensure that both flows will have similar number densities of nuclei available to the location of inception. However, very large changes of length scale may lead to a situation where the number density of available nuclei in the region of the lowest pressure changes significantly, making detection more difficult. Another implicit assumption of these equations is that screening effects immediately upstream of the point of cavitation inception are not important. Screening effects, which were identified by Johnson and Hsieh\cite{36}, could lead to a situation where nuclei bubbles of a certain size are forced away from the location of inception as the length scale changes. Screening would cause inception to be a function of different size nuclei bubbles. In effect, screening will change the nuclei distribution of the liquid and may cause equation 7.16 to be less accurate. As demonstrated by Johnson and Hsieh, these screening effects are important in cavitation inception along the surface of headforms, and is not expected to contribute to the dynamics of jet cavitation where nuclei bubbles are actually attracted toward the low pressure regions.

A final implicit assumption of equation 7.16 is that cavitation will occur immediately, regardless of the length scale of the flow. Some authors have hypothesized that the cavitation scale effects are a function of the time that cavitation nuclei spend in the region of a critical pressure (measured by an impulse function\cite{10}). The study of the nuclei bubbles dynamics shown in figure 7.2 reveals the time effects should only be important for high frequency fluctuations in the fluid. Since the cavitation in jet flows appears to be a consequence of coherent flow structures and not the random turbulent pressure fluctuations, the high frequency forcing should only play a part when the bubble is close to cavitating. Because of this, temporal effects are expected to be of secondary importance.

7.1.4 Effects of non-similar liquids

The scaling laws presented so far provide a means of predicting prototype cavitation numbers from model test data. These scaling laws only require that the two experiments are conducted in equivalent fluids which have the same (or very sim-
ilar) nuclei distributions. No information about the specific makeup of the nuclei bubbles is required. If, however, the nuclei distributions are well characterized, the equilibrium equation allows for even more general predictions. Medwin[69] measured the distribution of small bubbles in the coastal waters of California. These measurements showed that the size of nuclei bubbles changes very little from 5ft to 50ft below the ocean surface. The ambient pressure, however, will be quite different at these two depths (116,000 Pa at 5ft versus 250,000 Pa at 50ft). These differences in ambient pressure result in different initial pressures, \( p_0 \), in the bubbles. For a 100 \( \mu m \) bubble, the initial pressure, \( p_0 \), is found by setting \( a = 100 \mu m \) in equation 7.2 and letting the pressure, \( p \), be either 116,000 Pa at a depth of 5ft or 250,000 Pa at a depth of 50ft. Appropriate values of surface tension, \( S \), and vapor pressure \( p_v \) must be substituted as well. For this example, let these variables have values of \( S = 0.0728 N/m \) and \( p_v = 2300 Pa \). With these values, the initial pressure is found to be \( p_0 = 115,156 Pa \) at the 5ft depth and 249,156 Pa at the 50ft depth. These results show that \( p_0 \) will change significantly with depth below the ocean surface.

The initial pressure, \( p_0 \), is equivalent to a spring constant for the bubble. As the bubble changes size, \( p_0 \) determines the rate at which the bubble gas pressure changes. The question now is how much \( p_0 \) can affect the critical pressures at which cavitation would be expected (\( p_c \)). To answer this question, the values of \( p_0 \) can be substituted into equation 7.2 along with the definition of a cavitation event (\( a = 500 \mu m \) per Meyer et al.[10]). Substituting these values, equation 7.2 shows that the critical pressure will change from \( p_c = 2930 Pa \) at 5ft to \( p_c = 4002 Pa \) at 50ft, a 37% increase. While this change seems relatively small compared to the change of the background pressure, it will have an effect on the incipient cavitation number. Furthermore, an increase in the critical pressure is a detrimental change, meaning that the cavitation inception will occur sooner at greater depths.

Consider the jet flow from chapter 6 with a diameter of 0.1016 m (4 inches), a Reynolds number of \( Re = 500,000 \), and a jet velocity of \( U_J = 5.76 m/s \). Using \( C_{P_{min}} = -1.349 \) from the DES jet solution, the expected cavitation numbers for the 100 \( \mu m \) bubbles are \( \sigma_i = 1.387 \) at 5ft and \( \sigma_i = 1.452 \) at 50ft. The equilibrium cavitation theory predicts only a small increase in \( \sigma_i \) with depth. However, the results change considerably when a 1m jet is considered. In the larger jet, all of the
parameters will stay the same except for $U_J$, which will change to $U_J = 0.585m/s$. Using this velocity, the cavitation numbers change to $\sigma_i = 5.031$ at 5ft and $\sigma_i = 11.296$. The smaller velocity associated with the larger jet makes the first term in equation [7.11] much more significant. This, in turn, makes $\sigma_i$ more sensitive to depth for the larger jet.

The equilibrium theory also allows for predictions of how a change in nuclei size will affect the cavitation number. Using the previous data for a depth of 50ft, a 50$\mu$m bubble would have a larger interior pressure than the 100$\mu$m bubbles because of its larger surface tension. The magnitude of the initial pressure for the 50$\mu$m bubble would be $p_0 = 250,612Pa$, a slight increase over the initial pressure of the 100$\mu$m bubble (249,156Pa). The more significant change in equation [7.2] is the change in $a_0$. Since $a_0$ is raised to the third power, any change in this value will significantly alter the magnitude of the pressure within the bubble at cavitation inception. Changing both $p_0$ and $a_0$ for the 50$\mu$m bubble, the critical pressure for cavitation inception is 2260Pa. This value is less than the reported vapor pressure of water at 20$^\circ$C. Substituting this critical pressure into equation [7.11] give $\sigma_i = 1.347$ for a 0.1016$m$ diameter jet and $\sigma_i = 1.115$ for a 1$m$ jet. These represent significant reductions of $\sigma_i$ versus the values for the 100$\mu$m bubble at 50ft (1.452 for a 0.1016$m$ jet and 11.292 for a 1$m$ jet). The change is actually quite astounding for the 1$m$ diameter jet, dropping from $\sigma_i = 11.292$ to $\sigma_i = 1.115$ all because the nuclei bubble responsible for inception changed from 100$\mu$m in radius to 50$\mu$m in radius. This change in nuclei size also changes the relationship between $\sigma_i$ and $L_\infty$. When cavitation results from the The 100$\mu$m, $\sigma_i > -C_{P_{min}}$ and $\sigma_i$ increases with $L_\infty$. For the 50$\mu$m bubbles, $\sigma_i < -C_{P_{min}}$ and $\sigma_i$ decreases with $L_\infty$. This result shows that the cavitation number is highly dependent upon nuclei size and emphasizes that the nuclei distribution is a very important consideration for experiments. The equilibrium theory suggests that the liquid in test facilities should not be degassed and filtered to a point where the nuclei bubble distribution is significantly smaller than the nuclei bubble distribution which is expected in the prototype liquid. If the test liquid is degassed to remove all of the larger nuclei bubbles, the test results may predict a cavitation number which is significantly lower than the actual prototype cavitation number. This can lead to a situation where the test results provide a false sense of confidence that the prototype will
operate without cavitation within its design parameters.

As the previous example shows, the equilibrium scaling theory provides a possible explanation for some of the conflicting trends observed in experiments. We should expect that different experimental facilities will have different water supplies along with different methods of degassing and filtering the test liquids. Because of this, these facilities are likely to have different nuclei bubble distributions. The previous example of reducing initial nuclei sizes from $100\mu m$ to $50\mu m$ shows that it is not only possible for these facilities to observe different incipient cavitation numbers, but also entirely different trends with respect to length scale and Reynolds number.

### 7.2 Equilibrium Predictions of Jet Cavitation Inception

As demonstrated already, the equilibrium theory presented in section 7.1 can be used to make predictions of incipient cavitation numbers. Before we look at the numerical simulations of cavitation inception using the R-P equation and the EoM, let's use the equilibrium theory to make some predictions of the cavitation inception in the jet flow. Medwin\[69\] showed that naturally occurring nuclei in the oceans are commonly found with sizes up to about $100\mu m$ in radius. Meyer et al.\[10\] presents a summary of data from many different testing facilities which shows a similar distribution of nuclei. These sources of data suggest that nuclei between $10\mu m$ and $100\mu m$ in radius will be readily available in most water supplies. In an attempt to match this data, three nuclei sizes of $10\mu m$, $50\mu m$, and $100\mu m$ will be considered in the cavitation studies. The $100\mu m$ bubbles will be representative of “normal” or untreated water, while the $50\mu m$ and $10\mu m$ bubbles will be considered characteristic of highly degassed and filtered liquid sources.

The conditions considered in the numerical simulation include the assumption that the ambient far-field fluid pressure is $p_{\infty,0} = 101000 Pa$ (1 atmosphere), the vapor pressure is $p_v = 2300 Pa$, and the surface tension is $S = 0.0728 N/m$. These conditions are substituted into equations 4.47, 7.6, and 7.11 to obtain predictions for $100\mu m$, $50\mu m$, and $10\mu m$ nuclei bubbles. The results are summarized in ta-
ble \(7.1\) for a 0.1016\(m\) jet and in table \(7.2\) for a 1\(m\) jet. The minimum coefficient of pressure is obtained from the DES jet solution described in chapter \(6\). A search of the entire solution at each time step revealed that \(C_{P_{\text{min}}} = -1.349\). The results in table \(7.1\) show that for the smaller nozzle, all three bubble sizes result in cavitation numbers which are very close to \(-C_{P_{\text{min}}}\). This happens because the velocity of the jet is large, making the first term in equation \(7.11\) small. The larger nozzle has predicted cavitation numbers which are quite different from \(-C_{P_{\text{min}}}\). In this case, the velocity is much smaller which makes the first term in equation \(7.11\) much more significant. Recall that while the cavitation numbers change substantially with scale, the critical pressure where cavitation is predicted to occur remains constant for each bubble size. The large change in \(\sigma_i\) is a consequence of \(p_c \neq p_v\) and the change in velocity with length scale. In other words, the change in \(\sigma_i\) with \(D_J\) is a consequence of the definition of the cavitation number, and not due to a change in the physical pressure at which cavitation occurs.

<table>
<thead>
<tr>
<th>(D_J = 0.1016m)</th>
<th>(a_0)</th>
<th>(p_0)</th>
<th>(p_c)</th>
<th>(\sigma_i)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100(\mu m)</td>
<td>100156(Pa)</td>
<td>2810(Pa)</td>
<td>1.38</td>
<td></td>
</tr>
<tr>
<td>50(\mu m)</td>
<td>101612(Pa)</td>
<td>2110(Pa)</td>
<td>1.34</td>
<td></td>
</tr>
<tr>
<td>10(\mu m)</td>
<td>113260(Pa)</td>
<td>2009(Pa)</td>
<td>1.33</td>
<td></td>
</tr>
</tbody>
</table>

Table 7.1: Equilibrium predictions for the incipient cavitation numbers which result from various cavitation nuclei. The reference jet flow is assumed to have the following properties: \(D_J = 0.1016m\), \(U_J = 5.758m/s\), and \(C_{P_{\text{min}}} = -1.349\). The initial bubble pressure, \(p_0\), is found from equation \(4.47\), the critical pressure is calculated from equation \(7.6\) and \(\sigma_i\) is calculated from equation \(7.11\).

The ability of the equilibrium theory to predict cavitation numbers is dependent upon knowledge of the size of nuclei bubbles in the liquid and knowledge of \(C_{P_{\text{min}}}\). Both of these parameters can be difficult to obtain in practice. Recall from chapter \(6\) that probes placed in the simulated jet flow recorded a minimum \(C_P\) of only \(-0.82\), a value significantly higher than the actual minimum value of \(-1.349\). Table \(7.3\) shows that assuming \(C_{P_{\text{min}}} = -0.82\) will result in significantly different values of \(\sigma_i\) when compared to the assumption of \(-1.349\) listed in table \(7.2\). The only difference in these calculations of \(\sigma_i\) is the value of \(C_{P_{\text{min}}}\) entered into equation \(7.11\).
Table 7.2: Equilibrium predictions for the incipient cavitation numbers which result from various cavitation nuclei. The reference jet flow is assumed to have the following properties: $D_J = 1.0m$, $U_J = 0.585m/s$, and $C_{P_{min}} = -1.349$. The initial bubble pressure, $p_0$, is found from equation 4.47, the critical pressure is calculated from equation 7.6, and $\sigma_i$ is calculated from equation 7.11.

<table>
<thead>
<tr>
<th>$a_0$</th>
<th>$p_0$</th>
<th>$p_c$</th>
<th>$\sigma_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td>100$\mu$m</td>
<td>100156$Pa$</td>
<td>2810$Pa$</td>
<td>4.33</td>
</tr>
<tr>
<td>50$\mu$m</td>
<td>101612$Pa$</td>
<td>2110$Pa$</td>
<td>0.24</td>
</tr>
<tr>
<td>10$\mu$m</td>
<td>113260$Pa$</td>
<td>2009$Pa$</td>
<td>-0.35</td>
</tr>
</tbody>
</table>

Table 7.3: Equilibrium predictions of the incipient cavitation numbers using the smaller value of $C_{P_{min}}$ measured from the flow at hypothetical probes placed at $r/D_J = 0.5$. The value of $C_{P_{min}}$ measured from these probes is $C_{P_{min}} = -0.82$. The values of $\sigma_i$ are calculated from equation 7.11.

<table>
<thead>
<tr>
<th>$a_0$</th>
<th>$\sigma_i(0.1016m)$</th>
<th>$\sigma_i(1.0m)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>100$\mu$m</td>
<td>0.85</td>
<td>3.8</td>
</tr>
<tr>
<td>50$\mu$m</td>
<td>0.81</td>
<td>-0.29</td>
</tr>
<tr>
<td>10$\mu$m</td>
<td>0.80</td>
<td>-0.88</td>
</tr>
</tbody>
</table>

7.3 Numerical Test of Equilibrium Scaling Law

The last two sections have focused on cavitation scale effects by looking at the equilibrium form of the R-P equation. This approach provided scaling relations for $\sigma_i$ and resulted in a method to predict cavitation inception numbers for flows where both the largest nuclei bubble size and $C_{P_{min}}$ are known. However, the equilibrium approach neglects the dynamic terms of the R-P equation, terms which may become significant during the bubble growth associated with cavitation inception. The equilibrium theory also neglects any impact from the motions of the bubbles in the bulk liquid flow. Therefore, effects such as screening cannot be accounted for in the equilibrium theory.

In this section, the nuclei bubbles will be released into the jet flow presented in chapter 6. The nuclei bubbles will be allowed to respond to the flow according to an equation of motion (EoM) and the full R-P equation. The background pressure (cavitation number) is altered to determine the conditions when nuclei
bubbles would first become visible. The numerically determined values of $\sigma_i$ are then compared with various predictions from the equilibrium theory.

### 7.3.1 Operating Parameters

The details of the bubble dynamics code which numerically solves the EoM and R-P equations is discussed in chapter [5] This code is written to handle general bubble and particle dynamics and therefore has many input options. Table [7.4] reviews the input options used in the present bubble dynamics simulations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specified Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>R0</td>
<td>parametric value</td>
</tr>
<tr>
<td>n_lgn</td>
<td>1000</td>
</tr>
<tr>
<td>n_sub_tsteps</td>
<td>10000</td>
</tr>
<tr>
<td>delt</td>
<td>0.001</td>
</tr>
<tr>
<td>bubble_dynamics</td>
<td>.TRUE.</td>
</tr>
<tr>
<td>advection_scheme</td>
<td>‘‘rk4’’</td>
</tr>
<tr>
<td>re</td>
<td>500000</td>
</tr>
<tr>
<td>kinematic viscosity</td>
<td>$1.17 \times 10^{-6}$ (default)</td>
</tr>
<tr>
<td>density</td>
<td>1000.0 (default)</td>
</tr>
<tr>
<td>S</td>
<td>0.072 (default)</td>
</tr>
<tr>
<td>L_dim</td>
<td>parametric value</td>
</tr>
<tr>
<td>U_dim</td>
<td>$re*\text{kinematic viscosity}/L_{dim}$</td>
</tr>
<tr>
<td>kgas</td>
<td>1.0</td>
</tr>
<tr>
<td>sigma0</td>
<td>$(101000-2340)/(0.5<em>density</em>U_{dim}**2)$</td>
</tr>
<tr>
<td>sigma</td>
<td>parametric value</td>
</tr>
<tr>
<td>release_box</td>
<td>‘‘wedge’’</td>
</tr>
<tr>
<td>inlet_xmax</td>
<td>-0.1</td>
</tr>
<tr>
<td>inlet_xmin</td>
<td>-9.0</td>
</tr>
<tr>
<td>inlet_r1</td>
<td>0.0</td>
</tr>
<tr>
<td>inlet_r2</td>
<td>0.5</td>
</tr>
<tr>
<td>theta_1</td>
<td>0.0</td>
</tr>
<tr>
<td>theta_2</td>
<td>360.0</td>
</tr>
</tbody>
</table>

Table 7.4: Input parameters for the parametric study of cavitation inception.

The variables considered in the parametric study include the initial radius of the nuclei bubbles and the length scale of the flow. The bubble dynamics code solves the non-dimensional form of the R-P equation which was given by
Changing the length scale of the flow does not change the dimensionless dynamics of the bulk liquid flow, but does change the relative size of the nuclei bubbles and the dimensionless parameters in the EoM. Because of this, the response of the nuclei bubbles to the dimensionless fluid flow will be different as the length scale changes.

The parametric study is conducted by first choosing a length scale for the flow (i.e. choosing the jet diameter) and an initial bubble radius. Simulations are then conducted over a range of background pressures (cavitation numbers). The results of this first round of simulations are examined to determine which cavitation numbers resulted in cavitation events, and which did not. A second round of simulations are submitted with cavitation number in between the two limiting values found in the first round of study. The results of the second round are then examined to determine the cavitation numbers for a third round. This process is continued until the cavitation number is determined to the second decimal place.

Recall that the incipient value of the cavitation number corresponds to the largest $\sigma$ which results in at least 1 cavitating bubble ($a \geq 500 \mu m$). The parametric study revealed that the number of cavitating bubbles increases rapidly with a small decrease in $\sigma$ below the incipient value. Figure 7.3 demonstrates this rapid increase in events with decreasing $\sigma$.

The nuclei bubbles are initially placed randomly into a cylinder with the same diameter of the nozzle exit. This cylinder begins 9 diameters upstream of the nozzle exit and ends 0.1 diameter upstream of the nozzle exit. Bubbles within the inlet cylinder are advected at $\tilde{U} = 1$. As the bubble advect out of the inlet domain into the nozzle flow, they are set to their equilibrium radius and the velocity of the particle is set to the local fluid velocity. From this point on, the bubbles are governed by the full R-P equation and EoM. Tests were performed to determine a sufficient number of bubbles which would ensure detection of a cavitation event, with results summarized in table 7.5. In all cases, the cavitation number was set to the incipient value. The results indicate that the solution is independent of the number of the number of nuclei bubbles above a value of 10,000. However, smaller numbers of bubbles also predicted the cavitation number and actually showed a higher percentage of cavitation events. The following numerical simulations of $100 \mu m$ bubbles in $0.1016 m$ and $1.0 m$ jets were completed using 10,000 bubbles.
Figure 7.3: Percentage of 100µm nuclei bubbles which cavitate in the $D_J = 0.1016$ jet flow versus the cavitation number for the jet flow. All cavitation numbers larger than 1.38 resulted in zero cavitation events.

The computational costs of these parametric studies were too high relative to the amount of CPU time available for the project. To minimize computational expense, all other parametric studies were completed using only 1000 bubbles. Several spot checks were performed using 10,000 bubbles, all of which verified that 1000 bubbles were sufficient to detect cavitation inception.

<table>
<thead>
<tr>
<th>Number of Bubbles</th>
<th>Number of Cavitation Events</th>
<th>Percentage of Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,000</td>
<td>36</td>
<td>3.6 %</td>
</tr>
<tr>
<td>5,000</td>
<td>149</td>
<td>2.98 %</td>
</tr>
<tr>
<td>10,000</td>
<td>274</td>
<td>2.74 %</td>
</tr>
<tr>
<td>20,000</td>
<td>548</td>
<td>2.74 %</td>
</tr>
</tbody>
</table>

Table 7.5: Number of cavitation events versus the number of bubbles released into the inlet domain of the jet flow. All simulations were for a 0.1016m nozzle at $Re = 500,000$, $C_{P_{min}} = -1.349$, and $\sigma = 1.35$. 
Another input parameter which was investigated for sensitivity of the solution is the number of sub-time steps. To determine the appropriate number of sub-time steps to determine cavitation inception, 10,000 of the 100\(\mu\)m bubbles were released into the 0.1016\(m\) diameter jet. Table 7.6 summarizes the results and shows that 1000 sub-time steps are sufficient to capture the dynamics of the 100\(\mu\)m bubbles in this jet flow. However, the computational time associated with these numerical simulations is dominated by the time required to load the CFD data from disk, and not the time required to compute the bubble dynamics. Because of this, the number of sub-time steps was set to 10,000 for all of the parametric studies of cavitation inception.

<table>
<thead>
<tr>
<th>Number of Sub-Time Steps</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,000</td>
<td>277</td>
</tr>
<tr>
<td>5,000</td>
<td>274</td>
</tr>
<tr>
<td>10,000</td>
<td>274</td>
</tr>
<tr>
<td>15,000</td>
<td>275</td>
</tr>
<tr>
<td>20,000</td>
<td>278</td>
</tr>
</tbody>
</table>

Table 7.6: Number of cavitation events versus the number of sub-time steps used in the R-P and EoM dynamics. All simulations were for 10,000 bubbles of 100\(\mu\)m radius released in a flow with the following properties: \(D_j = 0.1016m\), \(Re = 500,000\), \(C_{P_{\text{min}}} = -1.349\), and \(\sigma = 1.35\).

### 7.3.2 Motion of Bubbles

It is not only the growth of the nuclei bubbles (and the resulting cavitation inception number) which is expected to change with the size of the nozzle. As discussed in chapter 3 and chapter 4, the parameters which govern the motion of the nuclei bubbles are also expected to change with the length scale of the flow. Recall the review of bubble motions in chapter 2, where the work of Ruetsch and Meiburg[42] was discussed. These authors showed that the rate of entrapment of bubbles by a vortical flow is a function of an inertia parameter, \(A\). For a bubble, the inertia parameter is defined as,

\[
A = \frac{9}{Re_f} \left( \frac{L_{\infty}}{a} \right)^2
\]
where $Re_J$ is the Reynolds number of the bulk liquid flow. The authors found that when $A$ was of order 1, or less, the bubbles were drawn into neighboring vortices. Bubbles with $A \gg 1$ tended to follow the bulk fluid velocity and were not drawn into vortices neighboring vortices. One difference between the present problem and the analysis of Ruetsch and Meiburg is that the size of the bubbles in the present work will be changing throughout the flow. This will cause the inertia parameter to change with location and $\sigma$. However, the bubbles will typically remain in the range of $10\mu m < a \leq 500\mu m$. Table 7.7 lists the values of $A$ for bubbles within this range for various jet diameters. The values of $A$ indicate that the smallest nuclei bubbles in the flow will follow the local fluid velocity closely, regardless of jet diameter. The larger bubbles are not so consistent. In the case of the 0.1016$m$ nozzle, as the bubble grow toward the size associated with cavitation, they will be drawn in toward the cores of any coherent vortex structures. However, as the size of the jet increases, vortices in the flow will exhibit less influence on the trajectory of the bubbles.

<table>
<thead>
<tr>
<th>$a_0$ (µm)</th>
<th>$L_\infty = 0.1016m$ (4 inch)</th>
<th>$L_\infty = 0.5m$</th>
<th>$L_\infty = 1.0m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1858</td>
<td>45000</td>
<td>$1.8 \times 10^5$</td>
</tr>
<tr>
<td>50</td>
<td>74</td>
<td>1800</td>
<td>7200</td>
</tr>
<tr>
<td>100</td>
<td>19</td>
<td>450</td>
<td>1800</td>
</tr>
<tr>
<td>500</td>
<td>0.74</td>
<td>18</td>
<td>72</td>
</tr>
</tbody>
</table>

Table 7.7: Values of the inertia parameter, $A$, for various bubble sizes and jet sizes ($Re_J = 500,000$).

Ruetsch and Meiburg also found that the percent of the total number of bubbles which become trapped in neighboring vortices is a function of the scaled particle settling velocity, $W$. For a bubble, the scaled settling velocity is,

$$ W = \frac{2}{9} \left( \frac{a^2 |\vec{g}|}{\nu U_\infty} \right). $$

(7.18)

The variable $\vec{g}$ represents the magnitude of the gravitation acceleration. $W$ will also change with location and $\sigma$. The percentage of the total bubbles captured by a vortex was found to increase with decreasing $W$. This happens because the magnitude of $W$ is proportional to the scaled velocity due to buoyancy. A large
value for $W$ indicates that the bubble buoyancy is strong and will overcome the influence that vortices have on the bubble. Small values of $W$ indicate that gravity has little influence on the particle, leaving only drag and inertia (described by $A$) to determine the trajectory. Values of $W$ are presented in table 7.8 for a range of bubble sizes and jet diameters. These results show that the bubbles are relatively unaffected by buoyancy, except in the case of the 1m nozzle. Even in this case, it is only the largest bubbles which are expected to experience a significant influence from buoyancy.

<table>
<thead>
<tr>
<th>$a_0$ (µm)</th>
<th>$L_\infty = 0.1016m$ (4 inch)</th>
<th>$L_\infty = 0.5m$</th>
<th>$L_\infty = 1.0m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>$3.2 \times 10^{-5}$</td>
<td>$1.6 \times 10^{-4}$</td>
<td>$3.2 \times 10^{-4}$</td>
</tr>
<tr>
<td>50</td>
<td>$8.1 \times 10^{-4}$</td>
<td>$4 \times 10^{-3}$</td>
<td>$8 \times 10^{-3}$</td>
</tr>
<tr>
<td>100</td>
<td>$3.2 \times 10^{-3}$</td>
<td>$1.6 \times 10^{-2}$</td>
<td>$3.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>500</td>
<td>$8.1 \times 10^{-2}$</td>
<td>0.4</td>
<td>0.8</td>
</tr>
</tbody>
</table>

Table 7.8: Values of the scaled particle settling velocity, $W$, for various bubble and jet sizes ($Re_J = 500,000$).

The values of $A$ and $W$ indicate that the motion of the bubbles will be affected by changes in $L_\infty$. Results of the numerical simulations for 10µm bubbles in different size jet flows are shown in figure 7.4. Both of these simulations are conducted at $\sigma$ slightly above $\sigma_i$. These results show that there is no qualitative difference in the distributions of the bubbles. This is expected since the values of $A$ are very large and $W$ very small for both nozzle sizes, indicating that the small bubbles will tend to follow the local fluid velocity. Results from simulations of 100µm bubbles are shown in figure 7.5 at three different jet diameters (note that figures 7.5a and 7.5c each have 10,000 bubbles while figure 7.5b only has 1000 bubbles). This figure shows significant qualitative differences between $D_J = 0.1016m$ and $D_J = 1.0m$. In this case, the values of $A$ for the large bubbles are of order 1 for the small jet flow, but order 100 to 1000 for the larger 1m jet. This indicates that the large bubbles in the small jet will be drawn into the cores of the vortex rings. Indeed, the bubbles show significant clustering in the cores of the rings in figure 7.5a. There is much less clustering of the bubbles in either of the larger flows, a result suggested by the larger values of $A$. The largest difference in these results is a consequence of buoyancy. The values of $W$ are small at $D_J = 0.1016m$,
increase to order 1 at $D_J = 1m$ since most of the bubbles have reached a radius of $a \sim 500\mu m$. Figure 7.5 confirms these expectations by showing almost no influence of buoyancy in either of the two smaller flows, but significant buoyancy effects in the large jet. All of these simulations are performed at $\sigma$ just above $\sigma_i$. For the $D_J = 1m$ flow, the ambient pressure is reduced significantly to cause cavitation inception. This reduction in background pressure causes all of the 100$\mu m$ bubbles, not just those near the low pressure regions, grow significantly. This forces all of the bubbles to experience significant buoyancy effects, effects which the bubbles would not normally experience at ambient conditions. To emphasize the size differences which lead to these solutions, the results are repeated in figure 7.6 at their relative size.

These results confirm that significant size effects are expected with the motions of the nuclei bubbles. Recall that these size effects are not accounted for in the equilibrium predictions of the previous sections. A concern now, is that these motion size effects may be important to the dynamics of cavitation inception, and have a significant impact on the applicability of the equilibrium approach to cavitation inception. Any impact on cavitation inception due to the motion of the nuclei will be captured in the numerical study of the full dynamics presented in the next section.

### 7.3.3 Cavitation Inception

The parametric study of cavitation inception consisted of guessing several cavitation numbers. The results of the numerical simulations at these cavitation number were examined for indications as to the value of $\sigma_i$. More simulations would be run at $\sigma$ value near the expected value of $\sigma_i$. These results were examined and this process was repeated until $\sigma_i$ was determined to 2 decimal places. In total, the parametric study of cavitation inception for 3 jet diameters and 3 nuclei sizes required nearly 100 numerical simulations for and more than 60,000 CPU hours. The data from the parametric studies is listed in appendix B. The incipient cavitation numbers which resulted from these studies are summarized in table 7.9 and show significant scale effects for $\sigma_i$. The values of $\sigma_i$ reported in table 7.9 represent the conditions required for at least 1 nuclei bubble to reach a radius of 500$\mu m$. 
Figure 7.4: Results of 10µm bubbles for (a) the 0.1016m nozzle flow at $\sigma = 1.2$ and (b) the 1.0m nozzle flow at $\sigma = -10.28$. 
Figure 7.5: Results of 100µm bubbles for (a) the 0.1016m nozzle flow at $\sigma = 1.4$, (b) the 0.5m nozzle flow at $\sigma = 2.1$, and (c) the 1.0m nozzle flow at $\sigma = 4.31$. 

(a) $D_J = 0.1016m$

(b) $D_J = 0.5m$

(c) $D_J = 1.0m$
Figure 7.6: Results of 100µm bubbles shown at relative size. The physical times from which the first bubbles left the nozzle are $t = 17.1s$ for $D_J = 1m$, $t = 4.3s$ for $D_J = 0.5m$, and $t = 0.18s$ for $D_J = 0.1016m$. 
Table 7.9 also lists the equilibrium predictions from section 7.2 in parentheses for comparison with the full numerical solutions. The 100$\mu$m and 50$\mu$m bubbles show very good agreement between the numerical simulations and the predictions from equilibrium theory. The 10$\mu$m bubbles, however, show significant discrepancies between the numerical simulations and equilibrium theory. These discrepancies are likely due to nonlinearities which are accentuated when the small bubbles grow almost an order of magnitude before cavitation inception occurs.

<table>
<thead>
<tr>
<th></th>
<th>$D_J = 0.1016 m$</th>
<th></th>
<th>$D_J = 1.0 m$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Numerical</td>
<td>Equilibrium</td>
<td>Numerical</td>
</tr>
<tr>
<td>$a = 100 \mu m$</td>
<td>1.38</td>
<td>1.38</td>
<td>4.30</td>
</tr>
<tr>
<td>$a = 50 \mu m$</td>
<td>1.32</td>
<td>1.34</td>
<td>0.21</td>
</tr>
<tr>
<td>$a = 10 \mu m$</td>
<td>1.19</td>
<td>1.33</td>
<td>-10.29</td>
</tr>
</tbody>
</table>

Table 7.9: Numerically obtained cavitation inception numbers compared with equilibrium predictions from tables 7.1 and 7.2.

While the equilibrium theory can be used to make predictions from an assumed nuclei size and $C_{P_{\text{min}}}$, the theory can also be used to consider how $\sigma_i$ will change from a model flow to a prototype flow. These predictions require no knowledge of the nuclei size, other than the two flows must have the same nuclei. Equation 7.16 is used with the results of the 0.1016$m$ jet to predict $\sigma_i$ for the 0.5$m$ and 1.0$m$ jets. These predictions are shown in table 7.10, the numerical results from table 7.9, listed in parentheses for comparison. The predicted values for the largest, most unstable nuclei bubbles (100$\mu m$), are nearly identical to the R-P solutions for both of the predicted length scales. For the smaller nuclei bubbles, equation 7.16 provides the proper trends, if not the same $\sigma_i$ value as the R-P simulations.

The equilibrium theory assumes that cavitation inception will occur at the location of $C_{P_{\text{min}}}$. To check this assumption, the numerically determined cavitation events can be plotting versus $x/D_J$ and dimensionless time. Figure 7.7 shows plots of the cavitation events at $\sigma_i$ and at a cavitation number just below $\sigma_i$ for the 0.1016$m$ jet. The corresponding 1$m$ jet cavitation events are shown in figure 7.8. Both of these figures indicate that cavitation inception is associated with the minimum pressure of the flow. Recall from chapter 6 that $C_{P_{\text{min}}}$ in the
D\_J = 0.1016m \quad D\_J = 1.0m

<table>
<thead>
<tr>
<th>a = 100\mu m</th>
<th>Numerical</th>
<th>Numerical</th>
<th>Predicted (eq 7.16)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.38</td>
<td>4.35</td>
<td>4.30</td>
<td></td>
</tr>
<tr>
<td>a = 50\mu m</td>
<td>1.32</td>
<td>-1.46</td>
<td>0.21</td>
</tr>
<tr>
<td>a = 10\mu m</td>
<td>1.19</td>
<td>-14.05</td>
<td>-10.29</td>
</tr>
</tbody>
</table>

Table 7.10: Comparison of the numerical solutions and the predictions given by the equilibrium scale effects from equation 7.16. The reference states for the equilibrium scale effects are the numerical results for \(D\_J = 0.1016\) listed in the second column.

CFD simulation was measured at \(x/D\_J = 4.5\). However, as cavitation number is decreased a small amount below \(\sigma\_i\), cavitation events appear elsewhere in the flow (i.e. \(x/D\_J = 2.5\)). This indicates that while \(C\_P\_min\) appears at only 1 time and space location, pressures elsewhere in the flow are of similar order of magnitude. In their experimental study of cavitation inception in a jet of the same Reynolds number, Ran and Katz[14] observed cavitation inception beginning at \(x/D\_J = 2.5\) and extending up to at least \(x/D\_J = 4\). While the present simulations indicate that inception will first occur at \(x/D\_J = 4.5\), a small decrease in \(\sigma\) shows that cavitation will quickly appear near \(x/D\_J = 2.5\). The present numerical simulations are predicting cavitation inception in the same location at which cavitation inception was observed by Ran and Katz.

### 7.4 Comparisons with Historical Data

Comparisons between equilibrium theory and full numerical simulations of the R-P equation show that both methods predict similar incipient cavitation numbers for the larger nuclei bubbles. However, the comparison of these two methods does not verify that the equilibrium scaling laws of section 7.1.3 can predict real flows. The equilibrium scaling relations must be tested with actual data. While most of the historical jet data does not provide measurements of \(C\_P\_min\), there is other data which can be compared with the equilibrium scaling theory.

Before looking at these other cases, it is still instructive to compare the numerical results for jet cavitation with the historical data. Figure 7.9 shows the present numerical cavitation results from the 100\mu m bubbles plotted with the historical
Figure 7.7: Cavitation events plotted versus axial location and dimensionless time for the 0.1016m jet. Cavitation inception is coincident with the occurrence of $C_{P_{\text{min}}}$ in the numerical simulation. While inception is found to occur at $x/D_J = 4.5$, a small reduction in $\sigma$ below the value of $\sigma_i$ initiates cavitation further upstream at $x/D_J = 2.5$. 

(a) $\sigma = 1.38(\sigma_i)$  
(b) $\sigma = 1.35$
Figure 7.8: Cavitation events plotted versus axial location and dimensionless time for the 1.0m jet. Cavitation inception occurs simultaneously at $x/D_J = 2.5$ and $x/D_J = 4.5$. The downstream location ($x/D_J = 4.5$) coincides with the occurrence of $C_{P_{min}}$. 

(a) $\sigma = 4.30(\sigma_i)$

(b) $\sigma = 4.27$
jet cavitation data originally discussed in chapter 2. The solid line in figure 7.9 represents the predictions from equation 7.16 using the 0.1016m results from the 100µm bubbles as the reference condition. The equilibrium theory and the numerical simulations of cavitation inception show strong scale effects at large jet diameters. For very small jets, the equilibrium theory predicts that the change in $\sigma_i$ with $D_J$ will be quite small and will asymptotically approach $-C_{P_{min}}$. However, $-C_{P_{min}}$ from the present study is much larger than the observed $\sigma_i$ values provided by the historical data. Furthermore, cavitation inception in the historical data is reported to occur further downstream ($x/D_J > 5$) than the observations in the present numerical study or the study by Ran and Katz[14].

![Graph showing cavitation inception numbers versus jet diameter](image)

Figure 7.9: Incipient cavitation numbers versus jet diameter. The solid line represents the equilibrium scale effects predicted by equation 7.16. The reference state is taken to be $\sigma_i = 1.38$ at $D_J = 0.1016$. This corresponds to the numerically determined incipient cavitation number from the 100µm parametric study.

The present CFD study and the experiments of Ran and Katz showed that the minimum pressure in a jet of $Re = 500,000$ will occur in the zone of flow establishment ($x/D_J < 6$). At the present time, there is still no conclusive argument to explain why visual cavitation is typically observed in the zone of established flow. A possible explanation is that the cores of the vortex rings generated by small jets
are of similar size as a visible nuclei bubble. The small size of the vortex rings may constrain the growth of the nuclei bubbles and prevent them from reaching a visible size (the pressure at the bubble surface increases as the bubble wall grows away from the vortex core[11]). Another possibility is that the bubbles begin to grow, but become unstable due to the relatively high shear in the small vortex rings. The combination of instability and shear then tear the bubbles apart and effectively creates a cap on the upper size of the bubbles, preventing them from being observed. A problem with these theories is that the pressure in the vortex cores is significantly lower than the pressure in the zone of established flow. This seems to indicate that the liquid in the vortex cores should turn to vapor long before the extremely low cavitation numbers needed to see inception farther downstream. However, vapor cavities were not witnessed in the near field of these experiments. It is possible that the initial growth of nuclei bubbles can alter the dynamics of the vortex rings. Sridhar and Katz[88] observed that even a few microscopic bubbles can deform the structure of a vortex. If the initial growth of bubbles in vortex rings can drastically alter their structure, $C_{P_{min}}$ would be changed from the observations of the present study. One would expect that the studies of Ran and Katz[14] or Gopalan and Katz[20] would have detected any significant change in vortex dynamics as cavitation numbers were reduced. It is possible, however, that their experiments were too large for this phenomena to be important. Another possibility is that the dynamics of these small jet flows is entirely different than the dynamics of the larger jets considered by Ran and Katz, Gopalan and Katz, and the present numerical simulations. Cerutti et al.[21] and Sridhar and Katz[88] have shown that the initial shear layer thickness of a jet is extremely important to the dynamics of the jet and the resulting value of $C_{P_{min}}$. Most of the older experimental studies do not provide details on the initial shear layer thickness and profile. They also make little or no mention of the unsteady dynamics of the jet flow. These studies provide only mean, and possibly root-mean-squared (RMS), values of pressure. Ran and Katz showed that the RMS values do not provide even a clue as to the $C_{P_{min}}$ values. Thus, the historical data is of little use for determining why cavitation is observed in the zone of established flow and not in the zone of flow establishment.

Because of the unknowns surrounding their cavitation inception, jet flows are
not ideal for verifying the equilibrium predictions. Better test cases are the NACA 16012 hydrofoil data and the 12% Joukowski hydrofoil data published by Holl and Wislicenus\cite{1, 2, 3}. Both hydrofoils were tested at 0° angle of attack. These hydrofoil data sets present a challenging test for the equilibrium theory. Not only were the tests conducted in different test facilities with different liquid qualities, but the tests show conflicting trends with both Reynolds number and length scale. Conflicting trends such as these have made it difficult to predict prototype cavitation numbers. One possible source of error for the equilibrium predictions arises in the determination of $C_{P_{\text{min}}}$ For both of these hydrofoils, reported value of $C_{P_{\text{min}}}$ is the theoretical value which is presumably obtained from inviscid calculations. This means that changes in $C_{P_{\text{min}}}$ with Reynolds number cannot be captured. Another problem is that Johnson and Hsieh showed that screening effects can occur in flows over headforms. While the hydrofoils have a streamlined shape, the stagnation point on the nose of the hydrofoil can lead to screening effects which may change with Reynolds number and length scale. Finally, the minimum coefficient of pressure will occur on the surface of the hydrofoil. Since nuclei bubble have finite size, there is a potential that the nuclei bubbles will not experience $C_{P_{\text{min}}}$.

As the length scale of the flow is changed, the relative size of the nuclei bubbles to the hydrofoil size will also change. This may change the value of the minimum coefficient of pressure which the nuclei bubbles experience.

The cavitation inception results for 2.5 inch and 5 inch chord length NACA 16012 hydrofoils tested at 0° angle of attack are shown in figure 7.10. This data shows that as the Reynolds number of the flow is increased, $\sigma_i$ is decreasing toward the value of $C_{P_{\text{min}}}$. The data also shows that $\sigma_i$ increases with the hydrofoil size. Equation 7.15 provides the expected change in $\sigma_i$ with Reynolds number if the length scale and liquid quality are held constant. The solid black line represents a curve defined by equation 7.15 with the reference state of $\sigma_i, Re_1 = 0.36$ at $Re_1 = 860000$. This line is then used as the reference state for equation 7.16 to predict the prototype cavitation inception values versus Reynolds number. The predictions given by equation 7.16 are shown as a blue dotted line. Considering that the actual value of $C_{P_{\text{min}}}$ is not known, the equilibrium scale effects equations do a remarkable job of predicting the value of $\sigma_i$ for the NACA 16012 hydrofoil data.

The data for the Joukowski hydrofoil was conducted as part of Parkin’s Ph.D.
thesis at The California Institute of Technology[1]. This data has been reproduced several other times as an example of prototypical cavitation scale effects[2, 3, 9]. Parkin’s experiments show that the incipient cavitation number for a Joukowski hydrofoil increases with Reynolds number, but decreased with an increase in the size of the hydrofoil. The data for the Joukowski hydrofoil is shown in figure 7.11. The trends for $\sigma_i$ are opposite of the trends for the NACA 16012 hydrofoil. This difference has been the subject of much speculation and debate since the flow geometries are not drastically different. Much of the speculation regarding the opposing trends has focused on differences in the surface profiles of $C_P$. The solid black lines in figure 7.11 are curves provided by the author which show the apparent trends of the data with Reynolds number. A problem with these trends is that they cross $-C_{P_{min}}$. The equilibrium scale effects theory (Equation 7.15) prohibits this from occurring unless the liquid quality has changed. Equation 7.15 shows us that when $\sigma_i = -C_{P_{min}}$, then it must always equal $-C_{P_{min}}$. However, when $\sigma_i \neq -C_{P_{min}}$, then it will never equal $-C_{P_{min}}$ as long as the liquid properties are held constant. The hydrofoil data in figure 7.11 is mostly less than $-C_{P_{min}}$. This
means that we should expect the value of \( \sigma_i \) to approach, but not equal, the value of \(-C_{P_{\text{min}}}\) as the Reynolds number increases. The fact that a few data points do cross the \(-C_{P_{\text{min}}}\) line can be attributed to experimental measurement error or many of the other sources of uncertainty which have been discussed.

![Cavitation data for 12% Joukowski hydrofoils at 0° angle of attack](image)

The theoretical \( C_{P_{\text{min}}} \) for this hydrofoil is \( C_{P_{\text{min}}} = -0.54 \).

A line defined by equation \((7.15)\) is drawn in red through the 2 inch chord length data in figure \(7.12\). The reference state for this line is \( \sigma_{i,Re_1} = 0.15 \) and \( Re_1 = 507,400 \). A similar line was drawn through the 4 inch-chord data (shown in blue) using a reference state of \( \sigma_{i,Re_1} = 0.0 \) and \( Re_1 = 1,072,000 \). These lines are used as reference states for equation \((7.16)\) to predict the incipient cavitation numbers for the 4 inch and 8 inch chord length data (shown as dotted lines). Predictions from the 2 inch curve (red) are shown as red dotted lines while the prediction from the 4 inch data (blue) is shown with a blue dotted line. Even though the trends with Reynolds number and length scale are opposite those of the NACA 16012 hydrofoil, the equilibrium theory predicts the proper scale effects associated with both parameters.

These two hydrofoil flows represented cases where \( \sigma_i \) exhibited opposing trends
with Reynolds number and size. The equilibrium theory was able to correctly predict the trends and the values of $\sigma_i$ for the larger geometries. The equilibrium theory also provides an explanation for the opposing trends of $\sigma_i$. The decrease in $\sigma_i$ with length scale for the Joukowski hydrofoil occurs because $\sigma_i < -C_{P_{\text{min}}}$. When $\sigma_i < -C_{P_{\text{min}}}$, $\sigma_i$ will decrease with an increase in length because the first term in equation 7.16 will be negative. The opposite trend will occur when $\sigma_i > -C_{P_{\text{min}}}$. This indicates that the occurrence of opposing trends is likely due to the liquid quality. If larger nuclei were available in the liquid used by Parkin, the opposite trends might have been observed. In general, when $\sigma_{i,m} < -C_{P_{\text{min}}}$, $\sigma_{i,p}$ will decrease as the length scale increases. When $\sigma_{i,m} > -C_{P_{\text{min}}}$, $\sigma_{i,p}$ will increase with the length scale. Regardless of the trends associated with length scale, the trend with Reynolds number will be for $\sigma_i \rightarrow -C_{P_{\text{min}}}$ as the Reynolds number increases. This will occur because the first term on the right-hand side
of equation \ref{eq:7.15} will approach zero as $Re_2 \to \infty$. As far as this author knows, the scale effects in these hydrofoil flows have not been predicted before. The equilibrium theory presented in this chapter appears to be a unique solution to predicting cavitation scale effects.
8.1 Summary

This thesis has investigated the scale effects associated with cavitation inception and explained that the cavitation scale effects in circular jets arises from the definition of the incipient cavitation number. The incipient cavitation number assumes that cavitation will occur when the minimum pressure in the flow reaches the vapor pressure. It was shown however that cavitation will actually occur when the minimum pressure reaches a value close to, but not equal to, the vapor pressure. The discrepancy between the actual pressure which causes cavitation and the assumed vapor pressure is the source of cavitation scale effects.

A dimensional analysis was performed to extract the significant dimensionless groups which govern cavitation inception in circular jets. It was shown that the incipient cavitation number depends on at least eight other dimensionless groups. Analysis of the dimensionless groups showed that they cannot all be held constant for a model and a prototype. These changes in \( \sigma_i \) with length scale were further investigated through a numerical simulation of cavitation inception. A numerical code was developed to simulate the response of cavitation nuclei in a fidelity detached-eddy simulation of a circular jet. The computational fluid dynamics simulation of the jet was shown to match jet flows reported in the literature. The
cavitation nuclei are governed by the Rayleigh-Plesset equation and the dispersion of the nuclei is governed by a semi-empirical equation of motion. A parametric study of cavitation inception confirmed that the incipient cavitation number changes as the size of the jet is changed. The simulations also showed that the initial nuclei size is very important for determining cavitation scale effects.

A scaling relation for cavitation inception was derived from the equilibrium Rayleigh-Plesset equation. This equilibrium scaling relation can be used to predict a prototype cavitation number based upon the model value of the incipient cavitation number. The equilibrium theory was validated by comparing with the numerical simulations and by comparing with historical hydrofoil data. Equilibrium predictions for 50\(\mu m\) and 100\(\mu m\) nuclei bubbles showed very good agreement with the numerical simulation for all jet diameters. The results indicate that the equilibrium scaling relation should be able to accurately predict the value of \(\sigma_i\) for hydrodynamic flows where nuclei bubbles with radii on the order of 100\(\mu m\) are common\[69,10\].

8.2 Conclusions

This thesis has investigated the origins of cavitation scale effects by examining the mathematical equations which describe the radial size of spherical bubbles in response to changes in the local fluid pressure. These bubbles serve as cavitation nucleation sites which are responsible for the first detectable occurrences of cavitation. The term “cavitation scale effects” is used to describe the deviation in \(\sigma_i\) from the expected value of,

\[
\sigma_i = -C_{P_{min}}. \tag{8.1}
\]

If equation 8.1 holds true, the value of the cavitation number for a prototype, \(\sigma_{i,p}\), should be predicted by a scale model. Unfortunately, experimental data often show \(\sigma_i \neq -C_{P_{min}}\) and also show significant scale effects (e.g. \(\sigma_{i,m} \neq \sigma_{i,p}\)). The dimensional analysis conducted in chapter 3 showed that cavitation is a function of many dimensionless parameters. The form of these dimensionless parameters makes it extremely difficult (if not impossible) to maintain similarity between a prototype and a model. It has also proved to be challenging to isolate the individual
dimensionless parameters to experimentally study the functional relationships of $\sigma_i$. This difficulty has motivated numerical studies of cavitation inception where the parameters of the problem are easily controlled. These numerical studies have shown that $\sigma_i$ is a strong function of the nuclei content of the liquid and shown that interactions of the bubble and the pressure field of the flow can lead to a selective filtering of nuclei bubbles (e.g. “screening effects” and “windows of opportunity”). Even with all of the experimental and numerical research into cavitation inception which has taken place over the last 50 years, the origins of cavitation scale effects have not been fully explained. A consequence is that it is still not possible to confidently predict the value of $\sigma_{i,p}$ for general problems.

In order to find an explanation for the origins of cavitation scale effects, the fundamental assumptions about the problem were investigated in chapter 7. The first assumption considered were those involved in equation 8.1. This equation assumes that cavitation inception will be detected at the instant the minimum pressure is reduced to the vapor pressure of the liquid. This assumes that the vapor pressure is the critical pressure required to cause cavitation inception. However, research has shown that hydrodynamic cavitation inception is not the result of a rapid change in phase of the fluid (homogeneous nucleation), but is the result of nuclei bubbles which change size in response to the pressures in the flow. The radial growth of these spherical nuclei bubbles is given by the Rayleigh-Plesset (R-P) equation which is a second-order nonlinear ordinary differential equation. Analysis of the R-P equation in chapter 7 showed that the equilibrium form of the R-P equation provides a close approximation of the full R-P equation until very close to the point where inception is expected to occur. While the equilibrium R-P equation is still nonlinear, it is an algebraic equation which is considerably simpler than the full R-P equation. The form of the equilibrium R-P equation is,

$$p_0 \left( \frac{a_0}{a} \right)^3 - \frac{2S}{a} = p - p_v.$$  \hspace{1cm} (8.2)

Using the equilibrium equation and an appropriate definition of a visible cavity ($a = a_{vis} = 500\mu m$), equation 8.2 was used to show that cavitation will be observed when the local pressure in the flow reaches pressures which are not necessarily equal to the vapor pressure. The critical pressure, $p_c$, which leads to observed cavitation
is given by,

\[ p_c = p_v + p_0 \left( \frac{a_0}{a_{vis}} \right)^3 - \frac{2S}{a_{vis}}. \]  

(8.3)

The initial pressure inside the bubbles, \( p_0 \), is given by,

\[ p_0 = p_{\infty,0} - p_v + \frac{2S}{a_0}, \]  

(8.4)

where \( p_{\infty,0} \) is the initial ambient pressure in the fluid and \( a_0 \) is the initial radius of the nuclei bubble at \( p_{\infty,0} \). Analysis of equation 8.3 shows that cavitation will only occur at the vapor pressure when,

\[ p_0 \left( \frac{a_0}{a_{vis}} \right)^3 = \frac{2S}{a_{vis}}. \]  

(8.5)

This condition is not guaranteed to be satisfied by the initial conditions.

Since \( p_c \) is not necessarily equal to \( p_v \), one of the fundamental assumptions of equation 8.1 is incorrect. Cavitation does not necessarily occur when \( p_{\text{min}} = p_v \), but rather when \( p_{\text{min}} = p_c \), where \( p_c \) is given by equation 8.3. To explore the implications of cavitation occurring at \( p_c \neq p_v \), the incipient cavitation number was re-written in terms of \( p_c \). The result was,

\[ \sigma_i = \frac{p_c - p_v}{\frac{1}{2} \rho U_{\infty}^2} - C_{P_{\text{min}}}. \]  

(8.6)

which shows that the classic view that \( \sigma_i = -C_{P_{\text{min}}} \) is only true when \( p_c = p_v \). In circumstances where \( p_c \neq p_v \), there must be a correction to equation 8.1. The correction is the first term on the right-hand side of equation 8.6. Chapter 7 explored the consequences of this further and showed that \( p_c \) will be a constant for tests conducted at different length scales as long as the liquid quality (nuclei distribution) and definition of cavitation inception are unchanged. Under these conditions, the numerator of the correction will be a constant. As the velocity of the flow is changed, the magnitude of the correction must change as \( U_{\infty}^{-2} \). Therefore the magnitude of the scale effects will change significantly as the velocity of the flow is altered. If dynamics similarity is not maintained as the scale changes, then the value of \( C_{P_{\text{min}}} \) will also be altered.

An equation for the prototype incipient cavitation number as a function of the
model incipient cavitation number was derived in chapter 7. This equation was derived by solving for \( p_c - p_v \) and then assuming that this quantity will remain constant. Setting the results for \( p_c - p_v \) for the model flow equal to the results for the prototype flow and solving for \( \sigma_{i,p} \) gave,

\[
\sigma_{i,p} = (\sigma_{i,m} + C_{P_{\text{min},m}}) \left( \frac{U_{\infty,m}}{U_{\infty,p}} \right)^2 - C_{P_{\text{min},p}}.
\]

Equation 8.7 shows how the correction term in equation 8.6 is expected to affect the incipient cavitation number as the conditions of the flow are altered. Some specialized limiting cases were examined as well. If the length scale of the flow is held constant but the Reynolds number is altered by changing the velocity, \( \sigma_i \) will change according to,

\[
\sigma_{i,Re_2} = (\sigma_{i,Re_1} + C_{P_{\text{min},Re_1}}) \left( \frac{Re_1}{Re_2} \right)^2 - C_{P_{\text{min},Re_2}}.
\]

Equation 8.8 shows that as the Reynolds number of the flow is increased, \( \sigma_i \) will asymptotically approach \( C_{P_{\text{min}}} \). This means that the correction term in equation 8.6 becomes less significant as the Reynolds number of the flow is increased. Equation 8.7 was also used to consider how \( \sigma_i \) will change between a dynamically similar model and prototype. For this problem, \( C_{P_{\text{min},m}} = C_{P_{\text{min},p}} \) and from Reynolds number similarity, \( U_{\infty,m}/U_{\infty,p} = L_{\infty,p}/L_{\infty,m} \). Using these relations, \( \sigma_{i,p} \) will be related to \( \sigma_{i,m} \) by,

\[
\sigma_{i,p} = (\sigma_{i,m} + C_{P_{\text{min}}}) \left( \frac{L_{\infty,p}}{L_{\infty,m}} \right)^2 - C_{P_{\text{min}}}.
\]

This equation shows that the correction to \( \sigma_i \) will actually be larger as \( L_\infty \) increases. This means that the error in equation 8.1 due to \( p_c \neq p_v \) will increase with the size of the flow.

Equations 8.7 through 8.9 show that the incipient cavitation number is expected to change with length scale - even when dynamic similarity is maintained. Errors to equation 8.1 occur because the definition of \( \sigma_i \) assumes that cavitation occurs at \( p_{\text{min}} = p_v \). When inception occurs at \( p_{\text{min}} = p_c \), where \( p_c \neq p_v \), then a correction
to equation 8.1 is needed to predict $\sigma_i$. The cavitation number and coefficient of pressure are both derived by non-dimensionalizing a physical pressure by the dynamic head of the flow. Since the difference between $p_c - p_v$ remains constant, the correction to equation 8.1 will change as the dynamic head of the flow changes. This change in the correction to equation 8.1 with the change in dynamic head is the origin of experimentally observed cavitation scale effects. While cavitation inception results from the same physical pressure in each test, the value of $\sigma_i$ will change as $U_\infty$ and $C_{P_{\text{min}}}$ change.

Regardless if cavitation inception is being predicted from first principles (equations 8.3 and 8.6) or from scale model tests (equations 8.7 or 8.9), the most important parameter of the problem is $C_{P_{\text{min}}}$. Without knowledge of $C_{P_{\text{min}}}$, accurate predictions of $\sigma_{i,p}$ cannot be obtained. Unfortunately, $C_{P_{\text{min}}}$ is difficult to measure, especially in unsteady separated flows. Without knowledge of $C_{P_{\text{min}}}$, it is not possible to determine how $\sigma_i$ will change with $L_\infty$ or Reynolds number. In these cases, computational fluid dynamics (CFD) simulations may be used to predict the value of $C_{P_{\text{min}}}$. Regardless whether $C_{P_{\text{min}}}$ is obtained from experiments or CFD, care must be taken to ensure that an accurate value is obtained. Errors in the measurement or prediction of $C_{P_{\text{min}}}$ will be compounded in predictions of $\sigma_{i,p}$.

The predictive abilities of the equilibrium scaling theory of equation 8.9 was also tested. Using the numerical results for a small jet (the model), the cavitation number for the larger jet (the prototype) was predicted. The agreement between equation 8.9 and the results of the parametric study for the 100$\mu$m bubbles is quite good. The agreement for the 50$\mu$m and 10$\mu$m bubbles gets progressively worse. Even though the discrepancies increased for the smaller bubbles, the correct trends in the change of $\sigma_i$ were always predicted. Since the largest commonly found bubbles are expected to cause cavitation inception, this theory appears to be appropriate for predicting cavitation scale effects. The exception would be for a liquid in which nuclei bubbles greater than 50$\mu$m were removed. In this case a full numerical simulation of the R-P equation would be required.

The comparison with the numerical simulations is promising. However, both the equilibrium theory and the numerical simulations are only approximations of reality. For the equilibrium theory to be confirmed, it must be able to predict cavitation scale effects for an actual flow. For this purpose, two previously unexplained
sets of cavitation inception data were examined with the equilibrium scaling theory. The data sets consisted of a NACA 16012 hydrofoil and a 12% Joukowski hydrofoil. While neither data set contained experimental measurements of $C_{F_{\text{min}}}$, both provided theoretical values. In both cases equation 8.8 was able to predict the change in $\sigma_i$ with Reynolds number. This is a significant accomplishment since the two flows showed different trends between $\sigma_i$ and Reynolds number. Equation 8.9 was used to predict the larger (prototype) inception numbers using only the curve from equation 8.8 which was fit through the model data. This equation was able to accurately predict the values of $\sigma_i$ for the larger prototype flows. Again, this is a significant accomplishment considering that the two data sets exhibit different trends between $\sigma_i$ and $L_\infty$.

In conclusion, this thesis has provided an explanation for the origin of cavitation scale effects and has derived a scaling formula which predicts cavitation inception in prototype flows. These equilibrium scaling formula and numerical simulations of cavitation inception predict that significant scale effects will exist for circular jet flows. Both methods predict that $\sigma_i$ will increase with $D_J$ for jet flows when cavitation occurs at a minimum pressure which satisfies the inequality, $p_{\text{min}} > p_v$.

### 8.3 Future Work

This thesis has derived equilibrium scaling formulas (equations 8.7 through 8.9) which predict how cavitation inception will scale with Reynolds number and length scale. While these formulas have demonstrated good agreement with the numerical simulations and limited hydrofoil data sets, they are still in need of more validation. A recommendation for future work would be to identify other sources of cavitation data which can be used to test the equilibrium theory. Data sets which represent other flow geometries would be especially desirable. Future work could also consist of the development of controlled validation experiments where $C_{F_{\text{min}}}$ is measured and the liquid quality can be controlled. The experiments should span both Reynolds numbers and length scale to ensure that equations 8.7 through 8.9 can be validated. Control of the liquid quality will allow the hypothesis that opposing trends with Reynolds number and length scale are explained by the magnitude of $p_c$ relative to $p_v$. 
The equilibrium theory was derived under the assumption that incipient cavitation is detected visually. Another means of detecting cavitation inception is with the sound waves generated when nuclei bubbles rapidly change size. This method of detecting cavitation is dependent upon rapid changes in the bubble volume, whereas visual detection only relies on the physical size of the bubble. It is possible that the critical pressure required to trigger acoustic emissions is more sensitive to frequency than the critical pressure required for visual observation. As the length scale of a fluid flow changes, so does its time scale (assuming dynamic similarity is maintained). The nonlinear dynamics of single bubbles should be investigated further to determine the conditions which lead to radiation of acoustic pressure waves. Once these conditions are identified, they should be compared with the assumptions in the equilibrium theory to determine if acoustically detected cavitation is expected to follow a similar scaling relation as visually detected cavitation.

The examination of the R-P equation in chapter 7 showed that the 100\(\mu\)m nuclei bubbles can be receptive to pressure fluctuations at frequencies which could correspond to the larger unresolved turbulent eddies (see figure 7.2a). Since the equations which govern the volume of the bubble are nonlinear, small pressure fluctuations can result in relatively large changes in the bubble radius if the pressure fluctuations have the correct frequency. It should be possible to develop a model for the sub-grid turbulent pressure fluctuations which are neglected by by combining a model for the magnitude of pressure fluctuations provided by Hinze\(^5\) with a model of a turbulent energy spectrum provided by Pope\(^{53}\). The model could be constructed by superimposing a finite set of sine waves which represent the pressure fluctuations associated with a band of wavenumbers. The magnitude of each sine wave would be set by Hinze’s model which is a function of the TKE. The TKE associated with each sine wave would be given by Pope’s model for the TKE spectrum. A model for the sub-grid turbulent pressure fluctuations could be used to assess the sensitivity of \(\sigma_i\) to turbulent pressure fluctuations.

Previous numerical studies of the interactions between bubbles and vortices have always assumed that the bubbles remain a constant size. This is an appropriate assumption for flows where the pressure drop in the core of the vortex will not lead to a large change in bubble radius (e.g. \(\sigma >> C_{P_{min}}\)). As the cavitation number of the flow is reduced, the nonlinearities of the bubble will eventually
lead to large changes in volume with only small changes in pressure. This could lead to significant changes in the inertia parameter, $A$, and the scaled particle settling velocity, $W$, which were shown by Ruetsch and Meiburg\[42\] to govern the interaction of bubbles and vortices. It may be possible to get a segregation where the bubbles outside of the vortex are unaffected by the vortex pressure gradients while bubbles near the vortex core are significantly affected. This could lead to a uniform distribution of bubbles outside of the vortex, a concentration of large bubbles at the vortex core, and no bubbles in between. Another possibility is that the bubbles inside the vortex will grow and be drawn into the core. As they continue to grow, buoyancy will become more important and the bubbles will try and rise out of the vortex ring. As they rise, they will experience an adverse pressure gradient, decrease in size, and lose the influence of buoyancy. This could lead to some interesting dynamics where the bubbles are biased to one portion of the vortex. Research into these dynamics could be conducted with the present bubble dynamics code. A suitable flow field would be a Stuart vortex which approximates vortices generated in a shear layer\[40, 89\].

The present simulations of cavitation inception in a jet suggests that cavitation will occur in the zone of flow establishment. This conclusion is supported by the studies of Ran and Katz\[14\] and Gopalan and Katz\[20\]. Most of the other experimental studies of cavitation inception in jets found that cavitation inception occurred in the zone of established flow ($x/D_j > 6$). The reason for these different locations of observation are not presently known. One possible reason as to why cavitation inception is observed in the zone of established flow is that the physical size of the jet experiments prohibits the growth of nuclei bubbles to a visible size. Hsiao et al.\[11\] have shown that in very tight vortices, the pressure gradients limit the growth of nuclei bubbles. It is possible that a similar phenomenon is occurring with small jets. Since small jets have small vortex rings, the physical size which bubbles can reach in these vortex rings is limited. It is possible to use the present CFD data and present bubble dynamics code to determine if small jets do limit the growth of nuclei bubbles. This can be performed with the help of the surface-averaged pressure (SAP) scheme proposed by Hsiao et al. This is an easy addition to the current code, but requires an additional 5 cell searches. The SAP scheme assumes that the pressure outside of the nuclei bubble is not the pressure
in the fluid at the location of the nuclei center, but instead is an average of the pressure at 6 polar locations on the bubble surface. This allows the pressure at the bubble wall to increase as the nuclei bubble grows larger and expands out from the center of a vortex. For large-scale flows where the vortex rings are much larger than a visible cavity, this scheme should make little difference. However, for small length scale flows where a visible cavity is a substantial portion of a vortex ring, the SAP scheme will limit the nuclei size to a more physically realistic value. It is important to note that one of the fundamental assumptions of the numerical scheme is that the bubbles are one-way coupled with the flow. In cases where the SAP scheme is necessary, this assumption is almost certainly violated. Sridhar and Katz[88] showed that nuclei bubbles which gather in the core of a vortex ring can significantly alter the dynamics of the vortex ring - even when the total void fraction of the bubbles is small. Under these conditions, a two-way coupled code would be needed to predict how the growing nuclei bubbles will alter the liquid flow. Regardless, it would be straightforward to use the existing code to check if small jets prevent bubbles from growing to a physically observable size in the zone of flow establishment. The only limitation on performing this study at the moment is the availability of computational resources.

Other interesting parameters to consider in future jet flows are swirl and co-flow. It is reasonable to expect that pumps may imparted some swirl on the jet flow in propulsion systems. This swirl will likely change the dynamics of the shear layer instability and the resulting formation of vortex rings. Since the minimum pressure occurs in response to vortex ring interactions, swirl has the potential to alter the cavitation performance of the jet flow. Jets in propulsion systems will also experience co-flow, or a background flow relative to the jet nozzle. As a ship is moving through the water, the ambient fluid is moving around the nozzle. Studies have shown that co-flow can significantly alter the location where cavitation is observed for a jet flow[16, 12]. These studies also showed that the incipient cavitation number was also altered. The dynamics which lead to the change in cavitation properties should be investigated with a numerical simulation.

The current jet simulation was conducted at $Re = 500,000$. This value is significantly lower than values which would be expected at full scale. The velocity for a jet with a diameter of $D_J = 0.5m$ and $Re = 500,000$ would have a velocity
of $U_J = 1.17m/s$ (2.3 knots). Considering that ship velocities are expected to be at least 10 knots, this velocity is much too small. If the jet were to operate at $U_J = 8m/s$ (15 knots) the Reynolds number would be $3.4 \times 10^6$. If the jet diameter were increased to $D_J = 1m$ and the velocity required to remain at $8m/s$, the Reynolds number would double to $6.8 \times 10^6$. These Reynolds numbers are much larger than what was considered in this study. It would be interesting to examine these much higher Reynolds numbers to determine what $\sigma_i$ might be in a more realistic full-scale jet.

The comparison of the DES and ILES results concluded that the DES solution was a better representation of the flow. However, the ILES simulation turned off the subgrid eddy viscosity in the entire flow domain. This caused the boundary layer inside of the nozzle to be much too thin (see figure 6.13). Perhaps a better way to conduct the comparison would be to use a RANS or DES subgrid turbulence model for the internal flow of the nozzle and only turn off the subgrid eddy viscosity for the separated flow downstream of the nozzle exit. This would ensure that the two flow start from the same conditions exiting the nozzle.
Octave FFT Script

The following script is an example of the commands used to calculate the fast Fourier transforms (FFT) of the transient jet data.

```octave
# gset term aqua;
# gset term postscript eps enhanced color;
axis([0.1,100,0.00001,1]);
gset xlabel ‘‘Dimensionless Frequency’’
gset size ratio 0.33
gset grid xtics ytics mxtics mytics

# for i=1:10
ii=i*10
variable= [‘‘fft_data.’’, int2str(ii), ‘‘.octave_data’’]
eval( sprintf( ‘‘load “%s”’’, variable))
u_des=u;
```
pr_des=pr;
U_DES=fft(u_des);
PR_DES=fft(pr_des);
variable=[‘fft_iles_r0p5.’,int2str(ii),‘.octave_data’]
eval( sprintf( ‘load “%s”’, variable))
u_iles=u;
pr_iles=pr;
U_ILES=fft(u_iles);
PR_ILES=fft(pr_iles);
variable2=[‘BOTH_fft_loglog_pr-x’,int2str(i),‘.eps’]
eval( sprintf( ‘gset output “%s”’, variable2))
variable3=[‘x/D_J = ’,int2str(i)]
eval( sprintf( ‘gset title “%s”’, variable3))
loglog(T_DES,abs(fftshift(PR_DES))/tlen,’1;DES;’,
T_ILES,abs(fftshift(PR_ILES))/tlen,’3;ILES;’)
endfor

The plotting capabilities of GNUPLOT are quite limited. The output from GNU-
PLOT is saved as encapsulated postscript (EPS) so that certain features of the plot can
be manually altered. The postscript files were modified by the following script which
which fixes the bounding box to remove unwanted white space. The script also modifies
the default line types to make them solid and bold.

#!/bin/bash
#
#
for i in 1 2 3 4 5 6 7 8 9 10; do
  name=‘BOTH_fft_loglog_pr-x’${i}‘.eps"
echo ${name}
sed -e ’s|BoundingBox: 50 50 410 302|BoundingBox: 50 45 410 190|’ 
    -e ‘s|{ PL|{ BL|’ 
    -e ‘s|2 dl 3 dl| ’ 

-e "'s|1 udl mul 2 udl mul| |'" \n- e "'s|BL { stroke userlinewidth 2 mul setlinewidth| \n    BL { stroke userlinewidth 4 mul setlinewidth|'" \n${name} > temp
mv temp ${name}
done
Appendix B

Parametric Cavitation Study

The following tables document the results of the parametric studies for cavitation inception. The operating parameters under which these tests were conducted are discussed in section [7.3.1]

<table>
<thead>
<tr>
<th>Cavitation Number ($\sigma$)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.30</td>
<td>1617</td>
</tr>
<tr>
<td>1.35</td>
<td>274</td>
</tr>
<tr>
<td>1.36</td>
<td>120</td>
</tr>
<tr>
<td>1.37</td>
<td>38</td>
</tr>
<tr>
<td>1.38</td>
<td>2 ($\sigma_i$)</td>
</tr>
<tr>
<td>1.40</td>
<td>0</td>
</tr>
<tr>
<td>1.50</td>
<td>0</td>
</tr>
<tr>
<td>2.00</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.1: Cavitation results for 10,000 nuclei bubbles with an initial radius of $a_0 = 100\mu m$. The diameter of the jet is $D_J = 0.1016m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{min}} = -1.349$. 
<table>
<thead>
<tr>
<th>Cavitation Number (σ)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.00</td>
<td>80</td>
</tr>
<tr>
<td>2.02</td>
<td>65</td>
</tr>
<tr>
<td>2.04</td>
<td>27</td>
</tr>
<tr>
<td>2.06</td>
<td>7</td>
</tr>
<tr>
<td>2.08</td>
<td>1 (σᵢ)</td>
</tr>
<tr>
<td>2.09</td>
<td>0</td>
</tr>
<tr>
<td>2.10</td>
<td>0</td>
</tr>
<tr>
<td>2.40</td>
<td>0</td>
</tr>
<tr>
<td>2.60</td>
<td>0</td>
</tr>
<tr>
<td>2.80</td>
<td>0</td>
</tr>
<tr>
<td>3.00</td>
<td>0</td>
</tr>
<tr>
<td>4.00</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.2: Cavitation results for 1,000 nuclei bubbles with an initial radius of $a₀ = 100 \mu m$. The diameter of the jet is $D_J = 0.5 m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{min}} = −1.349$.

<table>
<thead>
<tr>
<th>Cavitation Number (σ)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.00</td>
<td>10,000</td>
</tr>
<tr>
<td>3.00</td>
<td>9,604</td>
</tr>
<tr>
<td>3.50</td>
<td>1,572</td>
</tr>
<tr>
<td>4.00</td>
<td>321</td>
</tr>
<tr>
<td>4.25</td>
<td>178</td>
</tr>
<tr>
<td>4.27</td>
<td>107</td>
</tr>
<tr>
<td>4.29</td>
<td>24</td>
</tr>
<tr>
<td>4.30</td>
<td>3 (σᵢ)</td>
</tr>
<tr>
<td>4.31</td>
<td>0</td>
</tr>
<tr>
<td>4.35</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.3: Cavitation results for 10,000 nuclei bubbles with an initial radius of $a₀ = 100 \mu m$. The diameter of the jet is $D_J = 1.0 m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{min}} = −1.349$. 
<table>
<thead>
<tr>
<th>Cavitation Number (σ)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.15</td>
<td>155</td>
</tr>
<tr>
<td>1.20</td>
<td>140</td>
</tr>
<tr>
<td>1.25</td>
<td>95</td>
</tr>
<tr>
<td>1.28</td>
<td>34</td>
</tr>
<tr>
<td>1.30</td>
<td>4</td>
</tr>
<tr>
<td>1.31</td>
<td>3</td>
</tr>
<tr>
<td>1.32</td>
<td>1 (σ_i)</td>
</tr>
<tr>
<td>1.33</td>
<td>0</td>
</tr>
<tr>
<td>1.34</td>
<td>0</td>
</tr>
<tr>
<td>1.35</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.4: Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 50\mu m$. The diameter of the jet is $D_J = 0.1016m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$.

<table>
<thead>
<tr>
<th>Cavitation Number (σ)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1.00</td>
<td>771</td>
</tr>
<tr>
<td>-0.50</td>
<td>107</td>
</tr>
<tr>
<td>0.00</td>
<td>36</td>
</tr>
<tr>
<td>0.10</td>
<td>38</td>
</tr>
<tr>
<td>0.20</td>
<td>10</td>
</tr>
<tr>
<td>0.21</td>
<td>1 (σ_i)</td>
</tr>
<tr>
<td>0.22</td>
<td>0</td>
</tr>
<tr>
<td>0.24</td>
<td>0</td>
</tr>
<tr>
<td>0.26</td>
<td>0</td>
</tr>
<tr>
<td>0.28</td>
<td>0</td>
</tr>
<tr>
<td>0.30</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.5: Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 50\mu m$. The diameter of the jet is $D_J = 1.0m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$. 

<table>
<thead>
<tr>
<th>Cavitation Number ($\sigma$)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.50</td>
<td>130</td>
</tr>
<tr>
<td>0.80</td>
<td>58</td>
</tr>
<tr>
<td>1.00</td>
<td>25</td>
</tr>
<tr>
<td>1.10</td>
<td>19</td>
</tr>
<tr>
<td>1.12</td>
<td>13</td>
</tr>
<tr>
<td>1.14</td>
<td>7</td>
</tr>
<tr>
<td>1.16</td>
<td>2</td>
</tr>
<tr>
<td>1.18</td>
<td>2</td>
</tr>
<tr>
<td>1.19</td>
<td>2 ($\sigma_i$)</td>
</tr>
<tr>
<td>1.20</td>
<td>0</td>
</tr>
<tr>
<td>3.00</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.6: Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 10\mu m$. The diameter of the jet is $D_J = 0.1016m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$. 
<table>
<thead>
<tr>
<th>Cavitation Number (σ)</th>
<th>Number of Cavitation Events</th>
</tr>
</thead>
<tbody>
<tr>
<td>-20.00</td>
<td>All Deactivated</td>
</tr>
<tr>
<td>-15.00</td>
<td>All Deactivated</td>
</tr>
<tr>
<td>-14.00</td>
<td>All Deactivated</td>
</tr>
<tr>
<td>-13.00</td>
<td>All Deactivated</td>
</tr>
<tr>
<td>-12.00</td>
<td>All Deactivated</td>
</tr>
<tr>
<td>-11.00</td>
<td>114</td>
</tr>
<tr>
<td>-10.80</td>
<td>47</td>
</tr>
<tr>
<td>-10.60</td>
<td>22</td>
</tr>
<tr>
<td>-10.40</td>
<td>5</td>
</tr>
<tr>
<td>-10.30</td>
<td>3</td>
</tr>
<tr>
<td>-10.29</td>
<td>1 (σ_i)</td>
</tr>
<tr>
<td>-10.28</td>
<td>0</td>
</tr>
<tr>
<td>-10.26</td>
<td>0</td>
</tr>
<tr>
<td>-10.24</td>
<td>0</td>
</tr>
<tr>
<td>-10.22</td>
<td>0</td>
</tr>
<tr>
<td>-10.00</td>
<td>0</td>
</tr>
<tr>
<td>-5.00</td>
<td>0</td>
</tr>
</tbody>
</table>

Table B.7: Cavitation results for 1,000 nuclei bubbles with an initial radius of $a_0 = 10\mu m$. The diameter of the jet is $D_j = 1.0m$, the Reynolds number is $Re = 500,000$, and the minimum coefficient of pressure is $C_{P_{\text{min}}} = -1.349$. The term “All Deactivated” indicated that the bubble radius became unstable and $a \to \infty$. When this occurs, the EoM becomes unstable and the bubbles advect out of the domain (i.e. $\vec{x} \to \infty$). The code reports that the bubble has been deactivated.
Bibliography


Vita

Brian A. Edge

Brian Edge received his BS in mechanical engineering from The Pennsylvania State University, The Behrend College in May of 2000. Upon completing his undergraduate degree, Brian began work as a Product Development Engineer for Cummins Inc. in Jamestown, NY. At Cummins, Brian was responsible for design and development of valve trains, power trains, cylinder heads, and water pumps for the 11-liter ISM family and 16-liter ISX family of diesel engines.

In the fall of 2001, Brian resigned from Cummins Inc. to begin his MS degree in mechanical engineering at The Pennsylvania State University. Brian’s research consisted of numerical simulation of contaminant transport in the human thermal wake. This research was conducted at the Applied Research Laboratory (ARL) under Dr. Eric Paterson and in partnership with the Gas Dynamics Laboratory under the guidance of Dr. Gary Settles. This research led to a 2005 publication in the Journal of Fluids Engineering, volume 127, number 5. The article is entitled, “Computational study of the wake and contaminant transport of a walking human.” Brian completed his MS degree in the fall of 2003.

Brian began his Ph.D. at the Applied Research Laboratory in the spring of 2004. His Ph.D. research was generously funded by the Naval Sea Systems Command (NAVSEA). His Ph.D. work has led to a publication in the proceedings of the 26th Symposium on Naval Hydrodynamics which was held on 17-22 of September 2006 in Rome, Italy. The title of the conference paper is, “Modeling of cavitation inception in high-Reynolds number circular jets using detached-eddy simulation.” Upon completing his thesis in the spring of 2007, Brian will begin work at Knolls Atomic Power Laboratory in Niskayuna, NY.