A COMPUTATIONAL STUDY OF GASEOUS JETS
SUBMERGED IN A LIQUID CO-FLOW

A Thesis in
Aerospace Engineering

by

Melissa Fronzeo

© 2017 Melissa Fronzeo

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Master of Science

August 2017
The thesis of Melissa Fronzo was reviewed and approved* by the following:

Michael P Kinzel  
Research Associate  
Assistant Professor of Aerospace Engineering  
Thesis Advisor

Jules W Lindau  
Senior Research Associate  
Associate Professor of Aerospace Engineering

Zachary P Berger  
Research Associate  
Special Signatory

Joseph J Cor  
Research Associate  
Special Signatory

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

*Signatures are on file in the Graduate School
Abstract

The objective of this thesis is to apply Computational Fluid Dynamics (CFD) to multiphase flows to develop an understanding of the underlying flow mechanisms in artificially ventilated cavities and submerged gas jets. These studies are motivated by the need to better characterize these flows by understanding internal pressure and producing predictive regime mapping. The CFD is utilized to capture effects that are difficult to visualize experimentally.

First, a study on the internal pressure of artificially ventilated cavities is displayed. Investigation into both twin and toroidal vortex closure modes indicates that several pressure regions develop within the cavities. These regions are found to correlate to cavity expansion and contraction that are a function of the closure mode. It is found that the constant pressure assumption within the cavity, used within semi-empirical theory and prediction, is only accurate to a certain degree.

This thesis also contains several studies focused on multiphase submerged jets. First we examine the flow structure in the interaction of a gaseous jet with the ambient fluid tested at multiple densities. The main physical finding, consistent with previous observations, is that instabilities develop more rapidly at the gas-liquid interface with increased ambient fluid density. The dynamic response of the interface, partially driven by Rayleigh-Taylor and Kelvin-Helmholtz instabilities, drives other trends with increasing fluid density such as shorter Mach penetration and faster return to the ambient pressure. A breakdown of the jet spread and development is presented, along with the ambient fluid density effect on characteristics such as the size and spread of the mixing layer.

Lastly, submerged gaseous jets in a liquid co-flow at varying freestream velocities and mass flow rates of the jet, the controlling parameters, are studied. The results show that there are several distinct regimes that form with respect to the controlling parameters. These regimes are classified based on their shared time-averaged and dynamic characteristics, after which a regime map, similar to those used in multiphase pipe flow, is constructed. This regime map is generated with several different methods in order to find the best predictive measure of what regime will form.
## TABLE OF CONTENTS

List of Figures ................................................................. vii

List of Tables ................................................................. x

Acknowledgements ........................................................... xi

Chapter 1 Introduction ......................................................... 1
  Background and Motivation ................................................ 1
  Structure ................................................................. 3

Chapter 2
  Artificially Ventilated Cavities: Evaluating the Constant Pressure Approximation ....... 5
  Methods ................................................................. 6
    Numerical Methods .................................................. 6
    Geometric Configuration ............................................ 6
    Validation ........................................................... 7
    Methodology for Histogram Data Extraction ........................ 9
  Results ............................................................... 9
    Twin Vortex Cavity .................................................... 10
    Toroidal Vortex Cavity ............................................... 12
  Discussion ............................................................ 13
    Mixture of Gaussians on Pressure Histograms ........................ 13
    Pressure Variations and Resulting Cavity Flow .................... 18
    Pressure Variations and Semi-Empirical Theory .................... 21
  Conclusion ............................................................. 23

Chapter 3 An Investigation of Submerged Gas Jets ................................................. 24
  Methods ................................................................. 24
    Approach ............................................................ 24
    Validation .......................................................... 27
    Mesh Independence Study .......................................... 28
  Results ............................................................... 29
    Over-Expanded Submerged Jet (~ 0.2 NPR) ............................ 30
    Under-Expanded Submerged Jet (~ 0.08 NPR) .......................... 35
  Discussion ............................................................ 36
  Conclusions ............................................................ 40

Chapter 4 Gas Jets in a Liquid Co-Flow ......................................................... 41
  Methods ................................................................. 41
    Numerical Methods .................................................. 42
    Geometric Configuration ............................................ 42
    Validation ........................................................... 43
    Computational Mesh .................................................. 46
    Matrix of Conditions Explained ..................................... 47
  Results ............................................................... 48
LIST OF FIGURES

Figure 2-1: Geometry used throughout this work..........................................................7

Figure 2-2: Comparison of CFD predictions to the semi-empirical prediction of CQ’s........8

Figure 2-3: Left - Scaler cut out (z and y planes) showing the volume fraction of H2O gas in red. Right - The threshold containing cells in which the volume fraction of H2O gas is greater than 0.5..........................................................9

Figure 2-4: Sketch of Twin-Vortex Cavity taken from Semenko [1]; (a) Side View; (b) Top View..........................................................10

Figure 2-5: Sketch of Toroidal Vortex Cavity taken from Semenko [1]........................10

Figure 2-6: Representative pressure histogram highlighting the pressure distribution within a twin-vortex cavity and the regions that correspond to the identified pressure peaks......11

Figure 2-7: Representative pressure histogram of the pressure distribution within a twin-vortex cavity and corresponding regions........................................................................12

Figure 2-8: Representative PDF of the pressure distribution within a toroidal-vortex cavity and corresponding regions..........................................................13

Figure 2-9: Pressure histogram of Twin-Vortex Cavity with Mixture-of-Gaussians Applied..........................................................15

Figure 2-10: Pressure histogram of Toroidal-Vortex Cavity with Mixture-of-Gaussians Applied..........................................................17

Figure 2-11: Cavitation Number within a Twin-Vortex Cavity........................................19

Figure 2-12: Streamlines within a Twin-Vortex Cavity colored by the velocity magnitude and co-plotted with cavity pressure (greyscale)...........................................................19

Figure 2-13: Cavitation Number within Toroidal-Vortex Cavity ..................................20

Figure 2-14: Streamlines (colored by velocity magnitude) injected into Toroidal-Vortex Cavity co-plotted with the underlying pressure (greyscale)..........................................................21

Figure 2-15: Distribution of Cavitation Number in a Twin-Vortex Cavity ......................22

Figure 3-1: Diagram of the geometry used in these studies based on Ref. 5 .................26

Figure 3-2: Computational mesh in the near-jet region with contours indicating the local $M$..........................................................26

Figure 3-3: Comparison of (a) experimental snapshot to (b) CFD snapshot....................28

Figure 3-4: Comparison of the Centerline $M$ (NPR=0.2) with $\rho L=1$ kg/m$^3$ for the coarse, medium, and fine mesh ..........................................................29

Figure 3-5: $M$ damping ratio with refinement $\rho L=1$......................................................29
Figure 3-6: Jet-gas volume fraction ($\alpha_G$) at different $p_L$ values........................................31
Figure 3-7: M of the jet at different $p_L$ values.................................................................32
Figure 3-8: TVR plotted at different $p_L$ values.................................................................33
Figure 3-9: Pressure at different $p_L$ values.................................................................34
Figure 3-10: Combined density plots of nozzle M and LPR...............................................35
Figure 3-11: Volume fraction of gas at different $p_L$ values and 0.08 NPR..........................35
Figure 3-12: Combined density plots of nozzle M and LPR...............................................36
Figure 3-13: Figure indicating regions observed in the multiphase jet interaction..............37
Figure 3-14: Measurements of observed jet regions with variations in $p_L$ for the two nozzle conditions..................................................................................................................38
Figure 3-15: Change in the damping ratio with respect to $p_L$ (Extracted from Figure 3-10 and Figure 3-12).............................................................................................................39
Figure 3-16: Change in Amplitude/D$_{exit}$ with respect to $p_L$ (Extracted from Figure 3-10 and Figure 3-12).............................................................................................................39

Figure 4-1: Jet Geometry ..........................................................................................................43
Figure 4-2: Time Averaged Gas Location of Experiment and Mean Gas Volume Fraction of the CFD.................................................................................................................................45
Figure 4-3: Comparison of the time averaged gas location in Experiment (left) to the mean gas volume fraction from CFD (right) of the near jet region.................................................................46
Figure 4-4: Mesh Comparison ..................................................................................................47
Figure 4-5: Color contours indicate Mach number. Contour lines indicate the gas/liquid interface. The x-axis varies the jet mass flow. The y-axis varies the liquid velocity. These shots are all at the end of the simulation time.................................................................48
Figure 4-6: Overview of the observed regimes and general character.....................................52
Figure 4-7: $V_\infty = 1$ m/s, $m_{jet} = 0.0050$ kg/s, Time Sequence Figure..........................54
Figure 4-8: $V_\infty = 1$ m/s, $m_{jet} = 0.0050$ kg/s, Time Averaged.........................................55
Figure 4-9: $V_\infty = 1$ m/s, $m_{jet} = 0.0050$ kg/s, Snapshot.....................................................55
Figure 4-10: $V_\infty = 1$ m/s, $m_{jet} = 0.0050$ kg/s, Time Average..............................................56
Figure 4-11: $V_\infty = 10$ m/s, $m_{jet} = 0.0025$ kg/s, Time Sequencet.................................58
Figure 4-12: $V_\infty = 10$ m/s, $m_{jet} = 0.0025$ kg/s, Time Average.............................................59
Figure 4-13: $V_{\infty} = 10$ m/s, $m_{jet} = 0.0025$ kg/s, Snapshot ........................................59

Figure 4-14: $V_{\infty} = 10$ m/s, $m_{jet} = 0.0025$ kg/s, Time Average ........................................60

Figure 4-15: $V_{\infty} = 25$ m/s, $m_{jet} = 0.0050$ kg/s, Time Sequence ........................................62

Figure 4-16: $V_{\infty} = 25$ m/s, $m_{jet} = 0.0050$ kg/s, Time Average ........................................63

Figure 4-17: $V_{\infty} = 25$ m/s, $m_{jet} = 0.0050$ kg/s, Snapshot ........................................63

Figure 4-18: $V_{\infty} = 25$ m/s, $m_{jet} = 0.0050$ kg/s, Time Average ........................................64

Figure 4-19: $V_{\infty} = 10$ m/s, $m_{jet} = 0.0100$ kg/s, Time Sequence ........................................66

Figure 4-20: $V_{\infty} = 10$ m/s, $m_{jet} = 0.0100$ kg/s, Time Average ........................................67

Figure 4-21: $V_{\infty} = 10$ m/s, $m_{jet} = 0.0100$ kg/s, Snapshot ........................................67

Figure 4-22: $V_{\infty} = 10$ m/s, $m_{jet} = 0.0100$ kg/s, Time Average ........................................68

Figure 4-23: : $V_{\infty} = 5$ m/s, $m_{jet} = 0.0125$ kg/s, Time Sequence ........................................70

Figure 4-24: $V_{\infty} = 5$ m/s, $m_{jet} = 0.0125$ kg/s, Time Average ........................................71

Figure 4-25: $V_{\infty} = 5$ m/s, $m_{jet} = 0.0125$ kg/s, Snapshot ........................................71

Figure 4-26: $V_{\infty} = 5$ m/s, $m_{jet} = 0.0125$ kg/s, Time Average ........................................72

Figure 5-1: Time-Average Example of Each Regime ..................................................................75

Figure 5-2: Time-Averaged Matrices of Simulation Cases for Regime Classification (Images have been stretched in the y-axis) .................................................................76

Figure 5-3: Color Mapping of Assigned Jet Regimes ..............................................................77

Figure 5-4: 16x16 Linear Interpolation of ..................................................................................80

Figure 5-5: Filled 16x16 Linear Interpolation of .........................................................................80

Figure 5-6: 16x16 Cubic Interpolation of ....................................................................................81

Figure 5-7: Filled 16x16 Cubic Interpolation of .........................................................................81

Figure 5-8: Visual Separation and Mapping of .........................................................................82

Figure 5-9: Data Plotted with Equation 5.1 vs 5.2 ....................................................................83

Figure 5-10: Data Plotted with Equation 5.1 vs 5.2, Log-Log ..................................................84

Figure 5-11: Data Plotted with Equation 5.4 vs 5.5 ..................................................................85

Figure 5-12: Data Plotted with Equation 5.4 vs 5.5, Log-Log ..................................................85
LIST OF TABLES

Table 2-1: Corresponding Gaussian Parameters to Figure 2-9 .............................................................15
Table 2-2: Corresponding Gaussian Parameters to Figure 2-10 .............................................................17
Table 2-3: Application of Semi-Empirical Theory to Differing Cavitation Numbers .................23
Table 3-1: NPR measured in CFD compared to that specified ..............................................................30
Table 3-2: NEPR for each case .............................................................................................................30
Table 4-1: Comparison of Jet Characteristics from Weiland Experiment [4] and CFD ........45
Table 4-2: Percent Differences Between Experiment [4] and Simulation ............................................45
Table 5-1: Table of Numeric Regime Assignments ..............................................................................75
Table 5-2: Table of Case Regime Assignments ....................................................................................77
Acknowledgements

Funding was provided for Chapters 2, 4, and 5 through ONR under contract #N00014-16-1-2211, with Dr. David Drumheller as the program manager.

Funding for Chapter 3 was Funding provided through ONR under contracts #N00014-13-1-0692 and #N00014-16-1-2211, with Dr. Ronald Joslin as the program manager.

I would like to thank my fellow students and researchers Justin Valenti and Sam Hansford for both their moral support, as well their willingness to discuss my research and proofread my many document iterations.

I am very grateful for assistance of my signatories. Dr. Michael Kinzel is both my advisor and my mentor, and has been a constant source of input and assistance into my research since my undergraduate studies. Without his help, I never would have joined Penn State ARL and started a promising career in research. I would like to thank Dr. Jules Lindau for his input into my research and discussion of physics. I would like to thank both Dr. Zachary Berger and Dr. Joseph Cor for taking the time to read this thesis and provide input on their areas of expertise.

I am forever grateful for the constant love and support of my husband Jason Reeves, who has been a source of calm and patience throughout my studies.

Finally, I would like to thank my parents, Susan and Anthony Fronzeo, for their support in my studies, as well as the passion for knowledge that they cultivated in me.
Chapter 1

Introduction

Background and Motivation

Gaseous ventilated cavities forming within a liquid flow is a concept applied to many applications. Such applications include high-speed ships, drag reduction on hydrofoils, underwater cutting, and others [1,2]. These cavities form in the wake of a blunt object (or cavitator). They are formed either by directly injecting a non-condensable gas into the wake, or by entraining gas directly from the free surface. This type of cavitation is artificially ventilated cavitation. Such cavities are typically assumed to have a constant pressure within the cavity. This constant pressure is generally assumed the key characteristic of the cavity, from which theory [1] of the cavity can be fully established.

In a lateral, buoyant flow, these ventilated cavities are interesting in that they form multiple regimes. The two most common regimes are: (1) the twin-vortex regime, where a steady cavity closes with a pair of axially aligned, counter rotating vortices, and (2) toroidal vortex regime, where the cavity closes with an unsteady, single toroidal-shaped vortex. Each of these closure modes results in different cavity behaviors. It is reiterated that a key parameter to characterize these artificial cavities is the constant pressure within the cavity. Hence, artificial cavitation theory relies on the assumption that the pressure is constant throughout the cavity. In addition, potential flow methods rely on such a constant pressure assumption for boundary conditions. Furthermore, experiments often characterize cavitation results based on this parameter. Perhaps the only method that does not consider this assumption is CFD.

The constant pressure assumption has been largely uncontested until recent advances in the context of CFD. In the work of Kinzel (2009) [2], pressure variations within the supercavity was observed and suggested to play a role in the recovery of gas into the supercavity. More recently, Cao et al. (2017) [3] directly investigated pressure variations within gaseous cavities. This work observed vertical pressure
gradients, that look much like a hydrostatic pressure gradient as well as the pressure gradients at the closure point observed by Kinzel et al. (2009) [2]. Such pressure variations make sense from a fluid mechanics point of view, but are generally neglected in artificial cavitation analyses. The origin of this constant pressure assumption is probably due to the insensitivity of cavity-shape prediction to cavity pressure as well as difficulty measuring small pressure changes within a cavity. Chapter 2 evaluates the constant pressure assumption in the context of a computational fluid dynamics (CFD) model.

Chapters 3 and 4 shift focus to submerged gaseous jets. Again, this is a multiphase phenomenon; however, it now involves a supersonic injection of gas into the system. There are many studies of single-phase gas jets and gas jets expelled into another gas. In general, these types of jets are relatively well understood [4]. However, gas jets released into a dense ambient fluid or bubbly mixture, such as an air jet submerged into water, are not well understood. This lack of understanding is primarily due to the difficulties associated with experimental visualization. Such multiphase jets are dramatically different from their single-phase counterparts, and research continues to focus on developing the physical character and behavior of these gas-liquid jet flows. In the context of gas jets released into water, when below Mach 1, a submerged gas jet typically develops into a bubbly flow and is not very jet-like [4-6]. However, for gas speeds above sonic conditions, the gas flow transitions into a jet that is able to maintain its shape for a short distance. Such submerged jets are important for underwater cutting, underwater jet propulsion, nuclear reactor maintenance, and liquid metal mixing, among many other applications [5,6].

There are a number of existing experimental studies examining supersonic submerged gas jets out of the bubbly regime [4-7]. These results have led to the classification of several key differences between single-phase jets and multiphase jets. In this context, we refer to a multiphase jet as a compressible gas jet submerged within a bath of liquid. One characteristic phenomenon in these multiphase jets has been referred to as “back attack,” where shocks reflect off the gas-liquid interface and directly interact with the nozzle. This condition is associated with a bubbly flow, rapid expansion and collapse of large-scale bubbles, and localized bubble collapse on the nozzle surface itself (leading to nozzle damage) [5,6]. In the context of propulsion systems, back-attack buffets vehicles and is not desirable for vehicle control. In
addition, the gas-liquid interfaces are generally highly unstable. The high slip velocity between the gas jet and liquid flow leads to a jet flow with large-scale, coherent oscillations. Due to the highly unsteady nature and bubbly character of the flow, visual access in experimental setups is difficult. This has led to difficulty characterizing the gas-jet behavior interacting with a liquid.

An improved understanding of these multiphase jets can be explored using high-fidelity simulations tools such as computational fluid dynamics (CFD). CFD has been previously applied to examine submerged multiphase jets. In the context of compressible gas jets, the number of investigations is smaller. One previous work examined a submerged gas jet into liquid using an axisymmetric model [8]. The paper validated the CFD model using unsteady pressure near the nozzle exit that is created from shedding gaseous cavities. The validation indicated that CFD correlates well with the experimental measurements. Using this validated CFD model, they (with confidence) could conclude shock structure locations and characterize gas behavior. In addition, other work has validated CFD in the context of a compressible gas jets interacting with a gaseous liquid flow [2]. In Chapter 3, CFD is able to highlight difficult-to-measure interactions of the multi-species-gas-liquid flow. Based on these previous efforts, CFD indicates promise as a useful and critical tool to develop an understanding of the character of multiphase jets.

Structure

In Chapter 2, the pressure variations within a supercavity are documented based on results from a high-fidelity CFD method. The main objectives are to: (1) develop an improved understanding of pressure variations within an artificial cavity, (2) use these variations to understand general flow structure within the cavity, and (3) point out errors associated with these pressure measurements. To achieve these objectives, the paper begins by comparing the results of a CFD model with accepted semi-empirical theory to validate the model methodology. Using the validated model, the pressures in the supercavity are evaluated for several toroidal and twin-vortex type gaseous cavities. We then identify specific pressure
regions and how they relate to internal cavity flow. Finally, we develop an understanding of the impact of this assumption with respect to theory.

In Chapter 3, the aim is to develop the understanding of compressible multiphase jets using CFD. Results from higher-order, compressible-multiphase CFD with turbulence-resolving simulations are used to develop a physical understanding of the underlying mechanisms. The CFD is validated using available experimental data, which is limited to a qualitative validation of the bubbly structure. Combining this validation with mesh refinement studies, confidence in the predictions is developed. The predictions are then extended to examine the flow character of a converging-diverging nozzle jetting into quiescent liquid with varied densities, while maintaining a consistent pressure ratio at the nozzle outlet. These results are analyzed to develop insight into the mechanisms of the gas-jet interaction with various ambient fluids.

The purpose of Chapter 4 is to further explore these multiphase jets using CFD. The jet is modeled as a 2D Axisymmetric geometry with air being jetted into a bath of varying air-water mixture with an external liquid velocity. The geometry of the model is simplified in order to allow for a wider range of CFD data collection. The modeling is validated against experimental data by Weiland [4]. The effects of liquid velocity and mass flow rate of the jet are investigated with respect to the gas-liquid interactions. The jet formations are then studied in the context of regimes, in order to highlight the different formations of gas-liquid interaction for the multiphase jet. The internal flow of the resulting jet regimes are then investigated to provide further detail of the multiphase interactions.

Finally, the study in Chapter 5 builds off the data and regime classifications of Chapter 4 and works to produce a regime map, similar to those used for multiphase pipe flow. The regimes classified in Chapter 4 are assigned numeric values, and each case of CFD data is then placed in one of the resulting numeric regimes. These regime data points are investigated with respect to different methods of interpolation, in order to find the best method of finding regime transitions. In addition, two different sets of non-dimensional parameters are tested with the regime data to evaluate and in an attempt to isolate varying controlling parameters of the formations.
Chapter 2

Artificially Ventilated Cavities: Evaluating the Constant Pressure Approximation

The following chapter discusses the constant pressure assumption of artificially ventilated cavity correlations. The cavity pressure, $p_c$, in all correlations, is assumed constant throughout the cavity \cite{9}. The effect of this assumption reaches all predictive equations for these cavities, as it is a major parameter of the cavitation number, $\sigma$ (2.2), which in turn is used in drag and dimension predictions. The goal of this chapter is to evaluate this assumption and its effects on internal cavity flow, as well as cavity shape predictions.

The methods of the CFD simulation are discussed, after which the simulation results are analyzed. Local cavitation number CFD measurements within the cavity are binned, which results in several peaks, or regions of cavity pressure. These peaks are investigated with respect to corresponding regions in both Twin Vortex and Toroidal Vortex cavity closure regimes. The results show that there are consistent pressure regions, respective to each regime. A Mixture of Gaussians \cite{10} method is then applied to the cavitation number values binned into histograms, which involves modeling the peaks as Gaussian distributions. When these distributions are summed, they provide a reconstruction of the overall pressure distribution within the cavity.

Afterwards, the resultant pressure regions are investigated with respect to controlling the internal flow of the cavities, as well as how they affect the semi-empirical predictive equations. The results show that generally the constant pressure assumption holds, however there is a strong pressure gradient towards the closure regions of the cavities. This gradient may be a large driving force of the internal cavity flow structure.
Methods

Numerical Methods

All of the simulations were modeled using the commercial CFD code, Star-CCM+ [11]. In this work, a homogeneous, Eulerian-based multiphase-modeling approach that conserves phase mass through a volume-of-fluid-like formulation is used. Sharp interfaces are maintained using the HRIC scheme [12]. The CFD model is based on an incompressible, segregated-flow model that conserves mass, momentum, and energy. The numerical scheme is formally 2nd-order accurate in space and time. In terms of the turbulence, a $k - \epsilon$ turbulence model is used. The outer fluid is modeled as water, and the internal ventilated gas has properties similar to steam (but is noncondensable). The overall modeling approach has also been established and validated for similar flows in the context of supercavitation [2,13].

Geometric Configuration

The geometry used is of a disk cavitator. A diagram of the cavitator is provided in Figure 2-1. The disk has a flow-facing diameter of 0.08 m, a thickness of 0.0127 m (0.5 in), and an aft-facing diameter of 0.06 m. A cylindrical body is attached to the back of the cavitator and has a length of 0.2 m and a diameter of 0.02 m. Gas is injected into the cavity from the back face of the cavitator, minus the region where the body is attached. This gives an injection surface area of 0.0314 m$^2$. The cavitator is submerged in a bath of flowing water, shown in Figure 2-1. The dimensions of this domain are 10 m tall, 15 m long, and 5 m wide. The width is only 5 m as there is a symmetry plane that splits the domain and cavitator down the middle. The cavitator is placed in the bath 5 meters from the flow inlet, and is centered vertically (5 m from the top, 5 m from the bottom). The cavitator placement is shown in Figure 2-1.
Validation

Validation of the model is based on comparing CFD predictions to semi-empirical theory over a wide range of ventilation rates. In this effort, semi-empirical relations reviewed by Semenenko [1] are used to establish the CFD model validity. In Figure 2-2 is a prediction of the ventilation rate ($C_Q$) versus cavitation number ($\sigma$). The normalized ventilation rate, $C_Q$, is defined as

$$C_Q = \frac{Q}{D^2 V_\infty}$$  \hspace{1cm} (2.1)

where $Q$ is the ventilation rate, $D$ is the cavitation diameter, and $V_\infty$ is the free-stream velocity. The cavitation number, $\sigma$, is defined as:

$$\sigma = \frac{2(p_\infty - p_c)}{\rho V_\infty^2}$$  \hspace{1cm} (2.2)
where $p_c$ is the constant pressure within the cavity, while the infinity terms are the freestream properties. The comparison in Figure 2-2 shows that the CFD predictions correlate well with the $C_Q - \sigma$ empirical relation [1] with a Froude number ($Fr = V_\infty/\sqrt{gD}$) dependency.

$$C_Q = \frac{0.42C_{D_0}^2}{\sigma(\sigma^3Fr^4 - 2.5C_{D_0})}$$

(2.3)

$\sigma$ is also used in predictive empirical equations for cavitator drag, cavity length, and maximum cavity diameter. Using these equations to calculate $\sigma$ from simulation drag, diameter, and length, provided additional values for validation comparison.

\[ C_Q vs \ \sigma \ for \ Multiple \ Froude \ Numbers \]

\[ Fr = 19.3 \quad Fr = 14.6 \quad Fr = 11.0 \]

\[ C_Q = \frac{0.42C_{D_0}^2}{\sigma(\sigma^3Fr^4 - 2.5C_{D_0})} \]

\[ C_D = C_{D,0}(1 + \sigma) \]

\[ C_{D,0} = 0.82 \]

\[ (Disk \ Cavitator) \]

Increasing Pressure

Increasing Ventilation

Increasing Freestream Velocity

\[ Fr = \frac{V_\infty}{\sqrt{gD}} \]

\[ \sigma = \frac{2(p_\infty - p_c)}{\rho V_\infty^2} \]

Figure 2-2: Comparison of CFD predictions to the semi-empirical prediction of $C_Q$'s
Methodology for Histogram Data Extraction

In this effort, the focus is on the pressure behavior within the cavity regions. The first aspect of this involves isolating the solution to the gaseous cavity region. The cavity gas is isolated by thresholding the cells having a liquid-volume fraction less than 0.5 (essentially the interior of the cavity interface). An example of the isolated cavity region is given in Figure 2-3, where the red region of the left indicates a gas volume fraction of 1. The right side of the image shows the 3D threshold of cells, containing the entirety of the cavity. As the interface is thin, this threshold primarily contains the gas regions of the cavity. Within this gaseous subdomain, the cavity pressure (or $\sigma$) is sampled at every computational cell. These results are then evaluated using histograms, which have been normalized by the total number of cells, to provide the distribution of $\sigma$. Such an approach allows for a detailed interrogation of the pressure distribution within a cavity.

![Figure 2-3: Left - Scaler cut out (z and y planes) showing the volume fraction of H2O gas in red. Right - The threshold containing cells in which the volume fraction of H2O gas is greater than 0.5](image)

Results

The results of the pressure histograms are explored with respect to Toroidal and Twin vortex cavities. Twin vortex cavities are described as a moderate closure regime. They are longer cavities displaying a visible effect of gravity. Gas is ejected from the cavity from two hollow vortex tubes at the rear of the cavity, giving the regime its name [5]. A sketch of this regime is provided in Figure 2-4. Toroidal Vortex cavities occur when the effects of gravity are small (large cavitation numbers, large freestream velocities, large Fr) and the cavity can be considered axisymmetric. There is a reentrant jet at
the rear of the cavity where liquid flows back into the cavity, and gas is ejected from the cavity in toroidal vortices [5]. A sketch of this regime can be seen in Figure 2-5. In this section, the objective is to identify distinct cavity pressure regions using pressure histograms. Then, these regions are associated back to the physical regions of the cavities.

![Sketch of Twin-Vortex Cavity](image)

**Figure 2-4**: Sketch of Twin-Vortex Cavity taken from Semenko [1]; (a) Side View; (b) Top View

![Sketch of Toroidal Vortex Cavity](image)

**Figure 2-5**: Sketch of Toroidal Vortex Cavity taken from Semenko [1]

**Twin Vortex Cavity**

First consider a twin-vortex cavity at a Froude number of 14.6 and a ventilation rate of 0.65. For the twin vortex regime of cavities, review of the pressure histogram suggests that the cavity can be broken down into a series of peaks as depicted in Figure 2-6 (a). When the physical location of these peaks are highlighted within the cavity, as done in Figure 2-6 (b), it can be seen that they are associated with
specific regions of the cavity. The peak with the lowest cavitation number (Peak 1, highlighted in blue) is composed of a portion of the interface and the twin-vortex region of the cavity. The next peak, Peak 2 (highlighted in magenta), is the tallest peak (indicating the mode) and corresponds to the main body of the cavity. Peak 3, colored red, encapsulates the body behind the cavitator and forms a solid core to the front of the cavity, filling in a region of moderate radius growth. Lastly, Peak 4 is the smallest region with the highest $\sigma$ value (lowest pressure), which makes up the entirety of the area directly behind the cavitator, forming a cap around the ventilation area. Peak 4 also corresponds to the region with the most rapid radius growth. The spread in cavitation number across all 4 peaks is roughly 12%. These observations suggest that the twin vortex type cavity is divisible into regions of pressure that correspond to the cavity radius growth and shrinkage.

These aforementioned trends are repeatable. In Figure 2-7, a similar series of plots is provided at another twin-vortex cavity condition ($Fr=16.5$, $C_Q=0.5$), which has a cavitation number spread of about 15%. This type of histogram was observed for every twin vortex condition evaluated (from Figure 2-2). Hence, these four distinct regions in the supercavity are apparent beyond a single condition and appear to generalize to all twin vortex cavities with this modeling approach.

Figure 2-6: Representative pressure histogram highlighting the pressure distribution within a twin-vortex cavity and the regions that correspond to the identified pressure peaks.
Toroidal Vortex Cavity

The next type of cavity evaluated is a toroidal-vortex type cavity. For these cavities, the breakdown of pressure regions differs from that of the twin-vortex cavities. Results are plotted (in a similar form) in Figure 2-8. The largest peak for these cavities is observed to occur at the higher cavitation number (and thus lowest pressure). The pressure breakdown in the toroidal-vortex regime is radial rather than in the axial flow direction like with the twin-vortex regime. Peak 1 (highlighted in blue) appears to include the outermost interface of the cavity between gas and liquid as well as the most fore and aft portions of the cavity, as seen in Figure 2-8 (b). Peak 2 (magenta) forms a second thin shell layer on the interface and appears to capture more of the aft-cavity region. Peak 3 (red) continues to enclose the interface from the previous peaks and the aft region. Finally, Peak 4 (green) presents the mode and clearly dominates the cavity core. The change in cavitation number across all four peaks is roughly 4%. Similar to the twin-vortex cavities, results were verified to remain consistent for different conditions. Therefore, the pressure in these toroidal-shaped cavities tend to be dominated by the inner core, with regions that enclose the cavity as well as others that are associated with the reentrant jet.
Discussion

In the results section, it is shown that the constant pressure assumption within a ventilated, gaseous cavity remains mostly accurate, with the majority of the spread being in the fore and aft most portions of the cavity, with the bulk of the cavity being constant. However, cavities still form distinct regions of pressure, with cavitation number spread upwards of 15% across the distribution. Note that these regions are cavity dependent. Here we develop an understanding of the physical meaning of these variations.

Mixture of Gaussians on Pressure Histograms

To better isolate the peaks and pressure regions from the histograms of cavitation number, a Mixture of Gaussians (MOG) method was applied to better model the distributions. Such a method has been used in previous work to characterize cavitation clouds [10]. MOG assumes that the full distribution of the histogram is the sum of an unspecified number of Gaussian distributions [10]. In addition to
helping isolate distinct regions of pressure, it provides a probability function of the expected cavity pressure from a randomly selected pressure measurement.

The method involves finding the mean and standard deviations to match the slopes of the peaks. Then a weighting is applied to fit the peak height. When the individual Gaussian distributions, \( P_i(x) \), are summed, i.e,

\[
P(x) = \sum_{i=1}^{N} P_i(x),
\]

the resulting curve estimates the overall pressure histogram. The equation for each individual Gaussian distribution is as follows:

\[
P_i(x) = \frac{\phi_i}{\zeta_i \sqrt{2\pi}} \exp \left( \frac{(x - \omega_i)^2}{2\zeta_i^2} \right)
\]

Where \( \phi_i \) is a weighting factor, \( \zeta_i \) is the standard deviation, and \( \omega_i \) is the mean. The goal of the MOG analysis is for each of these \( P_i \) values corresponds to a different physical region to help guide the analysis to extract physical regions that span multiple peaks.

**Twin-Vortex**

Results from the MOG analysis for a twin-vortex cavity are displayed in Figure 2-9 and Table 2-1. This plot and table clearly highlight the observed peaks from the CFD model, and the combined curve well represents the CFD. Based on consistency of the MOG with the CFD observations, it is believed that the peaks can be generalized to four distinct regions:

- Peak 1 Region: Aft half of the bulk cavity (where the cavity radius is decreasing)
- Peak 2 Region: Fore half of the cavity (where the cavity radius is increasing)
- Peak 3 Region: Cavity growth region (where the cavity radius is moderately increasing)
- Peak 4 Region: Initial separation region (where the cavity radius is rapidly increasing)
In theory, using additional CFD cases, a MOG model can be empirically developed to better quantify pressure variations and regions within a twin-vortex cavity.

**Figure 2-9: Pressure histogram of Twin-Vortex Cavity with Mixture-of-Gaussians Applied**

<table>
<thead>
<tr>
<th>Distribution</th>
<th>Mean $\mu_I$</th>
<th>Standard Deviation $\sigma_I$</th>
<th>Weighting $\phi_I$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.07</td>
<td>0.0007</td>
<td>0.4</td>
</tr>
<tr>
<td>2</td>
<td>0.0718</td>
<td>0.0005</td>
<td>0.4</td>
</tr>
<tr>
<td>3</td>
<td>0.073</td>
<td>0.0004</td>
<td>0.08</td>
</tr>
<tr>
<td>4</td>
<td>0.0759</td>
<td>0.00066</td>
<td>0.12</td>
</tr>
</tbody>
</table>
**Toroidal Vortex**

Similar to the twin-vortex above, the MOG analysis is applied and results are presented for the toroidal-vortex cavity in Figure 2-10 and Table 2-2. In reconstructing this plot, four distinct peaks were found; however, the MOG analysis indicates that Peak 3 is associated with a distributed peak not observed in Figure 2-8 (b). The distributed behavior of Peak 3 can be observed in Figure 2-10. Because of this, the plots of Peaks 1 and 2, in Figure 2-8 (b), are highly likely to contain the content of the region defining Peak 3. Combining the results from the MOG with the observations from the CFD, it is believed that the peaks can be generalized to four regions:

- **Peak 1 Region**: Low pressures occurring during initial separation and closure.
- **Peak 2 Region**: Low pressures capturing more of the closure.
- **Peak 3 Region**: Transitional region from bulk cavity to the exterior flow and the reentrant jet.
- **Peak 4 Region**: Bulk cavity pressure of the cavity.

Similar to before, it is believed that using additional CFD cases, a MOG model can be developed to better quantify pressure variations and regions within a toroidal cavity as well.
Figure 2-10: Pressure histogram of Toroidal-Vortex Cavity with Mixture-of-Gaussians Applied

Table 2-2: Corresponding Gaussian Parameters to Figure 2-10

<table>
<thead>
<tr>
<th>Distribution</th>
<th>Mean $\bar{\sigma}_I$</th>
<th>Standard Deviation $\varsigma_I$</th>
<th>Weighting $\phi_I$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.1254</td>
<td>0.0003</td>
<td>0.07</td>
</tr>
<tr>
<td>2</td>
<td>0.1263</td>
<td>0.0004</td>
<td>0.04</td>
</tr>
<tr>
<td>3</td>
<td>0.1275</td>
<td>0.0015</td>
<td>0.2</td>
</tr>
<tr>
<td>4</td>
<td>0.12895</td>
<td>0.0004</td>
<td>0.3</td>
</tr>
</tbody>
</table>
Pressure Variations and Resulting Cavity Flow

These varying pressure regions correlate to cavity-radius growth and reduction, and appear to be the driving force of the internal, circulatory flow. Variation in pressure may seem small in these regions, however, the gas has a low density that can be responsive to these variations.

Twin-Vortex

Figure 2-11 depicts an overall axial pressure gradient within the cavity, where the darker regions indicate lower cavitation number (higher pressure) regions. In the case of the twin-vortex cavity, this gradient produces the lowest pressure just aft of the cavitator. From the cavitator to the tail, the cavitation number steadily decreases. This is likely associated with the cavity growth region causing an expansion, hence, lower pressure region up to the mid length of the cavity. When the cavity begins its closure, it causes a slight compression on the cavity that is likely to lead to a higher pressure. Note that the dominant vertical pressure gradient observed in the results of Cao et al.[3] is not observed in the present effort.

Additional evaluation of the internal flow within a twin-vortex cavity gives insight into the effect of the axial pressure gradient. In Figure 2-12 (a), a full twin-vortex cavity and its streamlines are plotted. It appears that a large portion of the flow attaches to the interface of the cavity and travels to the tail, where it is ejected through the twin vortices and is consistent with previous findings [2,3]. Additional examination of the internal flow in Figure 2-12 (b) shows the recirculating behavior within the cavity. It is apparent that the gas flow is responding to a positive pressure gradient in the tail and is redirected back into the cavity as a recirculating flow. A more detailed investigation of the flow is provided in Figure 2-12 (c), where only a few streamlines are displayed. From this flow visualization, it is apparent that the flow exposed to the high pressure gradient at the tail is repelled and slowly circulates through the cavity several times before finally being ejected through the tail. Hence, the results indicate that this overall axial pressure gradient drives the internal gas upstream within the cavity. This is fundamentally different from
the work of Cao et al. [3], which show a vertical pressure gradient. Such a vertical pressure gradient would imply a corresponding vertical gas flow. Such an inconsistency suggests a need for further work and potentially experiments to verify these findings. Nevertheless, it is believed that the present work refines the understanding of the pressure variations with respect to the cavity.

\[ \sigma \]

Increasing Pressure

**Figure 2-11**: Cavitation Number within a Twin-Vortex Cavity

(a) Full Opaque Streamlines
(b) Translucent Streamlines
(c) Reduced Streamlines

**Figure 2-12**: Streamlines within a Twin-Vortex Cavity colored by the velocity magnitude and co-plotted with cavity pressure (greyscale).
**Toroidal-Vortex**

A toroidal-vortex cavity is studied using a similar approach. In Figure 2-13 the pressure distribution is plotted along the center plane of the cavity. For this cavity type, the center of the cavity does not have the smooth transition of light-to-dark from the cavitator to the tail as the twin vortex did. Rather the center of the cavity contains a uniform bulk pressure that is consistent with the constant-pressure assumption. In addition, inside this region are high and low pressure pockets that correspond to liquid that periodically penetrates inside the cavity. A visible example of this is in Figure 2-13 where a high pressure (low cavitation number) band can be seen coming from the rear of the cavity into the center. At the closure region, there is a clear, dominate high pressure region. In addition, the pressure also remains high just aft of the cavitator. Nevertheless, the main difference observed in a toroidal cavity appears to be the lack of a constant axial pressure gradient. Similar to the how pressure differs between twin vortex and toroidal cavities, the gaseous flow also has several differences. An example of the predicted flow fields are plotted in Figure 2-14 (a) and (b). From the streamlines, it appears that the flow is recirculated through the cavity, cycling through the re-entrant portion of the cavity and slowly circulating downwards until it is ejected from the back of the cavity. As was the case in the twin-vortex cavity, the pressure gradients redirect and circulate flow within the cavity, with high pressure regions end capping the areas where the flow changes direction.

![Figure 2-13: Cavitation Number within Toroidal-Vortex Cavity](image-url)
Pressure Variations and Semi-Empirical Theory

The scaling parameter for ventilated cavity shape dimensions is the cavitation number [1,2]. This non-dimensional cavity pressure is the basis for other semi-empirical theory used to predict cavity dimensions and shape. Through these empirical correlations, the constant internal cavity pressure is invoked. This section aims to answer a simple question: When the pressure is evaluated at a point in the flow, how does the resulting cavitation number impact these relations? Such a concept correlates to an uncertainty associated with discretely sampling the pressure.

Consider the cavitation number histogram from one of the aforementioned twin-vortex cases, shown in Figure 2-15. It can be observed from the plot that the majority of the cavitation number values are within the range of $\sigma=0.066$ to $\sigma=0.078$. Taking several of these values, cavity dimensions and properties were calculated using the semi-empirical equations for cavitator drag, maximum cavity...
diameter, and cavity length. The results are presented below in Table 2-3. The values in Table 2-3 are generated using the semi-empirical theory reviewed by Semenenko [1] and are given as:

\[ C_D(\sigma) = C_{D0}(1 + \sigma) \]  
\[ D_c = D_n \sqrt{\frac{C_D}{k\sigma}} \]  
\[ L_C = \frac{A\sqrt{C_D}}{\sigma} \]

In these models, \( C_{D0} \) is 0.82 for a disk cavitator, \( k = 1 \), and \( A = 2 \). For the \( \sigma \)-values used, there is a 13% variation in cavitation number. In terms of the results from the semi-empirical relations, there is a 12% difference in total cavity length and a 6% difference in maximum cavity diameter. Drag differences appear to be minimal. Such differences indicate a modest effect of utilizing the constant-cavity-pressure assumption when evaluating the pressure, which should be considered.

Figure 2-15: Distribution of Cavitation Number in a Twin-Vortex Cavity
Table 2-3: Application of Semi-Empirical Theory to Differing Cavitation Numbers

<table>
<thead>
<tr>
<th>$\sigma$</th>
<th>$C_D$</th>
<th>$D_C$</th>
<th>$L_C$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.068</td>
<td>0.876</td>
<td>0.287</td>
<td>2.20</td>
</tr>
<tr>
<td>0.073</td>
<td>0.880</td>
<td>0.278</td>
<td>2.06</td>
</tr>
<tr>
<td>0.078</td>
<td>0.884</td>
<td>0.269</td>
<td>1.93</td>
</tr>
</tbody>
</table>

Conclusion

In the above sections, distinct regions of pressure, presented in terms of cavitation number, were observed. Though the constant pressure assumption holds in the bulk of the cavities (Peak 2 of the twin vortex cavity, Peak 4 of the toroidal vortex cavity) there is still significant spread across the histograms. The twin vortex cases showed a difference of up to 15% in the change across cavitation number, while the toroidal vortex cavity had a much lower spread. This indicates that the constant pressure assumption is better applied to toroidal vortex cavities. The findings of distinct pressure regions also indicate a high pressure region at the aft end of both cavity regimes examined. This high pressure region was investigated alongside predicted flow fields within the cavity, which indicates that this high pressure region forces flow to circulate back into the cavity. This internal flow investigation also highlighted the fact that the majority of the flow left the system along the interface of the cavity. The main importance of this chapter lies in semi-empirical theory. Experimentally, pressure is usually taken as a single probe. However, from the spread of values present, especially in twin vortex cavities, this can lead to upwards of a 10% difference in cavity dimension predictions. The overall implication is that the bulk of the cavity can typically be considered constant pressure, but the uncertainty of different pressure regions forming must be considered when applying the constant pressure approximation to cavity predictions.
Chapter 3

An Investigation of Submerged Gas Jets

The following chapter discusses variations in submerged gas jets. These jets are modeled as 3D, unsteady jets submerged in a large domain. The jets are air jetted into an external liquid with varying density. Additionally, the jets are simulated as both under-expanded and over-expanded, which affects the jet exit conditions such as exit wave formation.

The external liquid is modeled as water with densities of 1, 10, 100, and 1000 kg/m$^3$, in order to investigate the effect of external density on these jets. Results are compared between the different density cases, looking at pressure, Mach number, turbulent viscosity ratio, and volume fraction of the gas into the liquid. These results show increasing instability in the gas-liquid interface as the external density is increased, a finding that agrees with previous investigations of the different instabilities that affect submerged gas jets [5,6]. In addition to the interface breakdown, there is also an increased damping to the jet characteristics and waves, such as the jet Mach fluctuations at the exit.

These findings lead to the creation of a diagram that can be used to characterize the different sections of these submerged jets, including characterization of the core region, and layers of bubbly interface breakdown. These results can be used to help better characterize multi-phase submerged jets, which are one of the less studied jet formations [4-7].

Methods

Approach

In this work, a homogeneous, Eulerian-based multiphase-modeling approach that conserves phase mass through a volume-of-fluid-like formulation is used. Using the commercial CFD code, Star-CCM+ [11], sharp interfaces are maintained using the HRIC scheme [12]. The CFD model is
based on a compressible pressure-based, segregated-flow model that conserves mass, momentum, and energy. The method is not based on a conservative form, thus for compressible multiphase flow some modeling errors are expected (and assumed to be relatively small). The numerical scheme is formally 2nd-order accurate in space and time. In each case, the jet gas is modeled as a compressible gas based on air fluid properties. The ambient fluid (henceforth denoted as liquid) is modeled as incompressible. All of the liquid fluid properties are based on water; however, the constant water density, \( \rho_L \), is used as a test parameter. In terms of the turbulence modeling approach, a hybrid Reynolds Averaged Navier-Stokes (RANS)/Large Eddy Simulation (LES) turbulence model is used. The model is based on the Spalart-Allmaras Delayed-Detached Eddy Simulation (DDES) formulation [15]. The usage of such a DDES approach is based on previous work that suggests such formulations provide reasonable accuracy in the interaction of a gas-liquid interface with slip [2]. Thus, the present work employs DDES as a model that ad-hoc alleviates interfacial turbulence model errors, and does not claim to be resolving a significant portion of the inertial range of the turbulence (which is a goal as the work evolves). The overall modeling approach has also been established and validated for similar flows in the context of compressible gas jets interacting with a gas/liquid flow [14].

Now consider the experimental setup that was modeled. It contains a converging-diverging nozzle from the experiment of Shi et. al [5]. A diagram of the nozzle geometry is provided in Figure 3-1. The nozzle has a throat diameter of 4.3 mm and an exit diameter of 5.6 mm. The nozzle is placed into a liquid bath and is positioned on the left side of the bath shown in Figure 3-1. Within this bath is the “ambient fluid” or liquid. The bath dimensions are 1 m x 1m x 2.75 m. The bath-domain boundaries are modeled as slip walls. The only exception is the top boundary, which is modeled as a pressure outlet boundary set to the ambient pressure. This bath domain is a reasonable representation of the experimental setup that is also quite large (with respect to the jet size), hence it is expected to represent the experiment and a general submerged gas jet.
The aforementioned geometry is then discretized into a computational mesh. The computational mesh is based on a hex-dominant, unstructured mesh with prism layers within the nozzle boundary layer. The mesh is generated using the Star-CCM+ mesher [11]. An overview of the mesh is depicted in Figure 3-2, which also plots the corresponding Mach ($M$) contours (with red being the highest Mach and blue being the lowest). The mesh is refined in the near-jet region that generally remains inviscid; the higher resolution is used in this region to reduce dissipation through shock structures. The resolution of this refined-mesh region is around $0.1 \ D_{exit}$, which is rather coarse for the single-phase limit, but is sufficient to start understanding trends in the dynamics. An abrupt coarsening region occurs downstream of the refinement zone, which tends to diffuse the far-jet predictions when an inviscid core extends to that distance.
The focus of the present study is to develop an understanding of the effect of ambient-fluid densities for a gas jet submerged in liquid. An investigation of other parameters, such as compressibility and viscosity, is provided in Appendix A, which are found to be of less importance to this interaction. The following investigation involves a matrix of eight conditions that varies both \( \rho_L \) and the exit nozzle condition. The value of \( \rho_L \), the ambient fluid density, was varied from 1 to 10 to 100 to and to 1000 kg/m\(^3\). To consistently compare these different liquid densities, the nozzle pressure ratio (NPR) was matched for all cases. NPR is defined as the freestream pressure divided by the stagnation pressure at the nozzle inlet \( (p_\infty/p_0) \). The effect of \( \rho_L \) is evaluated for two NPR values. The NPR values include both an over-expanded case \( (\text{NPR}=0.2, p_{\text{exit}} < p_\infty, \text{shock at exit}) \) and an under-expanded case \( (\text{NPR}=0.08, p_{\text{exit}} > p_\infty, \text{expansion waves at exit}) \). Thus, the multiphase jet interaction initiates with either expansion or compression waves, respectively. The combined studies are designed to evaluate the effect of exterior density under various nozzle exit conditions to provide a generalized understanding of the interaction.

**Validation**

The experiments of Shi et al [5] are used to validate the CFD model. The nozzle geometry and stagnation pressures specified from the experiment are matched in the CFD model. In this case, an NPR of 0.107 is considered. A comparison between snapshots from the experiments and the CFD simulations provide a qualitative validation of the CFD model results. This comparison is provided in Figure 3-3. Figure 3-3 (a) provides experimental photographs while Figure 3-3 (b) provides snapshots from the CFD that indicate instances that a similar character is observed in CFD model. The similar character includes large-scale flow oscillations in the bubble regions in the entire jet (indicated by time-varying jet radius), “back-attack” as can be inferred from the variable cloudy regions near the nozzle, and very similar jet dissipation character. The lack of availability of additional data limits us to this sort of qualitative comparison. Although such a validation process is not fully compelling, the same numerical scheme was validated using experiments of a gas jet interacting with a gas-liquid flow [5,6]. This
qualitative validation, combined with previous validation work for a similar flow, provides confidence that the present CFD scheme can provide insight when studying this phenomenon.

Mesh Independence Study

In order to verify that discretization error is not dominating the solution, a sensitivity study to the mesh and time resolution was performed. For this study, an under-expanded multiphase jet with a $\rho_L$ value of 1 kg/m$^3$ is considered. This low-$\rho_L$ condition is examined as it maintains the longest shock structure, which is the worst-case scenario for the discretization error. Three mesh resolutions were considered, which resulted in free-jet-mesh resolutions of $\Delta x/D_{exit} = 0.1, 0.2, \text{ and } 0.4$. For each of these resolutions, the local $M$ along the jet centerline is plotted versus the axial length from the jet exit plane in Figure 3-4. The coarse-mesh clearly indicates that the shock structure disappears altogether. With decreased cell size, the overall jet length and structure approaches a consistent result. This conjecture is verified in a process where, from Figure 3-4, we extract an effective damping ratio of the centerline $M$. Such a behavior correlates to damping in the free jet, which should directly relate to numerical dissipation. This damping ratio is plotted, as a function of mesh resolution, in Figure 3-5. From this plot it is clear that as the mesh is refined, this damping ratio is convergent, indicating that the results are asymptotically converging to the bulk properties of the jet. The estimated numerical uncertainty of this
prediction is 0.022. The overall conclusion from this study is that, although the fine-scale turbulent-jet structures are not fully resolved, the overall influence of the jet and ambient fluid interaction is captured.

Figure 3-4: Comparison of the Centerline $M$ (NPR=0.2) with $\rho_L=1 \text{ kg/m}^3$ for the coarse, medium, and fine mesh

Figure 3-5: $M$ damping ratio with refinement $\rho_L=1$

Results

To achieve the over- and under-expanded jet-exit conditions the value of $NPR$ is fixed for all $\rho_L$ values. As indicated in Table 3-1, there are small discrepancies in $NPR$ due to the fact that it is a result of the solution. Note that the iterative process arises due to modeling the facility. All eight cases are within 10% of the prescribed value. Table 3-2 provides the nozzle exit pressure ratio ($NEPR$), which is the pressure at the exit of the nozzle, divided by $p_\infty$ (i.e. $p_{exit}/p_\infty$). This ratio provides a clear measure over the jet exit condition, i.e., over-expanded if $p_{exit}/p_\infty < 1$, or under-expanded $p_{exit}/p_\infty > 1$. These eight cases are designed to evaluate the impact of $\rho_L$ for the various jet-exit conditions. From the CFD model, investigations are focused on the turbulent viscosity ratio (TVR), jet-gas volume fraction ($\alpha_G$), Mach number ($M$), and pressure ($p$). These parameters, respectively, indicate turbulent breakup mechanisms, jet-gas diffusion, jet-speed reduction, and the redistribution of jet energy.
Table 3-1: NPR measured in CFD compared to that specified

<table>
<thead>
<tr>
<th>Target Ratio</th>
<th>$\rho_L=1$ kg/m$^3$</th>
<th>$\rho_L=10$ kg/m$^3$</th>
<th>$\rho_L=100$ kg/m$^3$</th>
<th>$\rho_L=1000$ kg/m$^3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Over-Expanded Jet</td>
<td>0.2001</td>
<td>0.2007</td>
<td>0.2006</td>
<td>0.1985</td>
</tr>
<tr>
<td>NPR=0.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Under-Expanded Jet</td>
<td>0.07995</td>
<td>0.08229</td>
<td>0.08131</td>
<td>0.08716</td>
</tr>
<tr>
<td>NPR=0.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3-2: NEPR for each case

<table>
<thead>
<tr>
<th>Target Ratio</th>
<th>$\rho_L=1$ kg/m$^3$</th>
<th>$\rho_L=10$ kg/m$^3$</th>
<th>$\rho_L=100$ kg/m$^3$</th>
<th>$\rho_L=1000$ kg/m$^3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Over-Expanded Jet</td>
<td>0.4971</td>
<td>0.4869</td>
<td>0.5374</td>
<td>0.5368</td>
</tr>
<tr>
<td>NPR=0.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Under-Expanded Jet</td>
<td>1.351</td>
<td>1.249</td>
<td>1.133</td>
<td>1.119</td>
</tr>
<tr>
<td>NPR=0.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Over-Expanded Submerged Jet (~ 0.2 NPR)**

First, consider the impact of $\rho_L$ for the over-expanded jet conditions. A contour plot of the jet-gas volume fraction ($\alpha_G$) is provided on a slice through the center of the jet in Figure 3-6. Note that $\alpha_G$ indicates the local content, by volume, of the jet gas. The red regions directly indicate regions dominated by gas. The extent of $\alpha_G$ indicates the external jet character, while the internal values indicate the local density, distribution of gas, and the dissipation of gas into the liquid. The results clearly indicate a reduction of gas-jet penetration and an increase of gas-jet diffusion with a denser surrounding liquid.

Figure 3-6 (a) indicates that for low $\rho_L$ values, the jet remains intact until its disruption where the mesh coarsens. As $\rho_L$ increases to 10 kg/m$^3$, plotted in Figure 3-6 (b), the jet appears to breakdown and diffuse into the liquid after roughly $x/D_{exit} = 18$ lengths downstream of the nozzle exit. For a $\rho_L$ value of 100 kg/m$^3$, plotted in Figure 3-6 (c), the jet breakdown occurs at $x/D_{exit} = 2$ and has difficulties penetrating the liquid. For the largest $\rho_L$ value of 1000 kg/m$^3$, which is consistent with water, the result is plotted in
Figure 3-6 (d) and the jet breakdown occurs immediately. In addition, the signs of “back-attack” are apparent as observed from the gas interacting with the jet surface. Unsteady video of Figure 3-6 (d) indicates an unsteady pulsation of the gas jet, both in the jet-surface interaction and the formation of large bubbles that are shed from the jet region.

The local Mach number \( M \), based on the gas sound speed, is plotted near the jet in Figure 3-7. Note that \( M \), as plotted, only accounts for the acoustic speed of the gas and does not consider effects of mixtures that reduce the sound speed. These plots are used to highlight the character of the gas-jet speed. Similar to the previous plot, Figure 3-7 (a) indicates the result for the low \( \rho_L \) value (of 1 kg/m\(^3\)). In this condition, the jet persists for a long distance and processes through several shock/expansion events. As \( \rho_L \) is increased to 10 kg/m\(^3\) (plotted in Figure 3-7 (b)) the shock structure appears dampened slightly compared to the \( \rho_L = 1 \) kg/m\(^3\) condition. This is indicated by the shrinking of the high-Mach, red region in the core of the jet. As \( \rho_L \) increases further to values of 100 and 1000 kg/m\(^3\), plotted in Figure 3-7 (c) and (d) respectively, the jet dampening continues to increase and the high-speed core flow is significantly shorter (i.e., the jet penetrates the high-density liquid much less than a low-density liquid). These contour plots of Mach number indicate a strong reduction of speed and increased dampening of the jet as the ambient density is increased.
The dampening of the jet is a feature that typically coincides with dissipation processes. For this reason, we investigate turbulence viscosity ratio, TVR. TVR is defined as the ratio of the sub-grid-scale modeled turbulent viscosity to the molecular viscosity of the gas/liquid mixture. In this work, we neglect the resolved turbulent scales. When TVR is large the diffusion terms in the RANS equations are large, which can directly dampen fluid-flow features. TVR is plotted using contour plots on the center plane of the jet in Fig. 8. Again, increasing $\rho_L$ values are plotted for Figure 3-8 (a) through (d). For the low-density case, the turbulence levels are relatively low. However, for the max-$\rho_L$ case, in Figure 3-8 (d), the turbulence is very high, with dense pockets of high TVR. In addition, the spread of the non-zero TVR values greatly increase, corresponding to the spread of the gas into the domain. This is witnessed in the large cloud of TVR visible in Figure 3-8 (d). Such turbulence is likely excited through Kelvin-Helmholtz instabilities at the gas-liquid interface. In general, these plots indicate that the modeled turbulence tends to correlate directly to the jet damping and breakdown.
Lastly, pressure contours are plotted on the jet centerline in Figure 3-9. The subfigures of Figure 3-9 indicate increasing $\rho_L$ values, where $\rho_L = 1$, 10, 100, and 1000 kg/m$^3$ align with Figure 3-9 (a), (b), (c), and (d), respectively. Similar to the previous plots, these images clearly indicate the low and high-pressure regions that occur in the shock cells of the supersonic jet. The length of these pulsations shrinks as $\rho_L$ increases, which is consistent with the previous observations. When interrogating the pressure (in regions away from the shock structure) large-scale, mild pressure fluctuations can be observed in the flow. These are indicated by the dark-blue regions that appear to grow in size as $\rho_L$ increases in Figure 3-9 (c) and (d). When compared to $\alpha_G$, in Fig. 6, the right-most pressure rise corresponds to the region near the leading edge of the gas bulge from the jet. It appears that the gas bubble acts like a solid body that accelerates the liquid, causing a pressure rise. In Figure 3-9 (d), there is another mild pressure rise that occurs between the shedding and growing cavities. Smaller versions of these pressure oscillations are observed in Figure 3-9 (c). The conclusion from these pressure plots is that as the gas bubbles shed, they accelerate, which drives a pressure gradient throughout a large region of the flow. This indicates that turbulent mixing is not the only mechanism driving the liquid flow.
Quantitative comparisons that highlight the effect of $\rho_L$ can be drawn from plots of $M$ and local pressure ratio (LPR) in Figure 3-10. Note that the LPR is defined as the ratio of pressure to stagnation pressure in the nozzle. In these plots, the time averaged parameters are extracted from a streamline passing through the center of the nozzle (the streamline is used to account for buoyancy). The axial distance, $x$, is plotted from -50 mm to 50 mm, with $x = 0$ corresponding to the jet exit. The nozzle throat is located at $x = -10$ mm. These plots are used to enable more precise comparisons.

In examining the Mach number versus axial distance plot in Figure 3-10 (a), it can be observed that the spatially averaged $M$ remains relatively consistent for $\rho_L$ values of 1 and 10 kg/m$^3$. For the $\rho_L$ of 100 kg/m$^3$, the oscillations begin to break down after about two. For a $\rho_L$ value of 1000 kg/m$^3$, the $M$ drops rapidly after the first oscillation. In addition, despite maintaining a supersonic jet flow, there are discrepancies between each $\rho_L$ value just inside the nozzle. Such an upstream effect may indicate that liquid affects the nozzle flow through shear-layer mechanisms. Now consider the LPR values plotted along the same streamline in Figure 3-10 (b). This plot clearly indicates that the dampening characteristics observed from the decay rate of the LPR amplitude directly correlate to $\rho_L$. This observation is consistent with the previous qualitative observations. These observations may provide insight into nozzle damage, jet penetration, and jet acoustics in various media.
Under-Expanded Submerged Jet (~ 0.08 NPR)

The next case considered is the nozzle in an under-expanded condition with an NPR value of 0.08. The physical characteristics of the jet are highlighted through a contour plot of $\alpha_G$ provided in Fig. 11. The trends are very similar to those observed for the over-expanded jet in Figure 3-6. Perhaps the main difference can be observed in Figure 3-11 (c), where for a $\rho_L$ value of 100 kg/m$^3$, the jet immediately mixes with the liquid. When comparing Figure 3-11(c) to Figure 3-6 (c), the jet retained its shape for a shorter length prior to breaking down. This may indicate that the jet type influences the near-nozzle stability.
It was found that the remaining flow-field plots indicated very similar results to the over-expanded condition. For this reason, results are not directly discussed in the main text, but are provided in Appendix B for reference. The general conclusion is that the exit flow character appears to affect instabilities occurring in the near-nozzle flow region.

Similar to before, the plots along the streamline passing through the center of the nozzle are provided in Figure 3-12. Figure 3-12 (a) plots the time averaged $M$ versus axial distance, while Figure 3-12 (b) plots time averaged LPR versus axial distance. The results are consistent with the over-expanded nozzle. This consistency includes the dramatic drop in $M$ for the $\rho_L = 1000 \text{ kg/m}^3$ condition (with respect to the lower $\rho_L$ values), slight modification to the results inside the nozzle, and an increased dampening with increased $\rho_L$. Thus, the impact of the liquid density to the bulk effects of the jet flow appears to be relatively independent of the jet condition.

**Discussion**

The CFD indicates the expected trends, that with an increased liquid density, the jet becomes shorter. This initial characterization focuses on the length where various processes occur in the jet. In addition, insight into the underlying flow behavior was also elucidated. A diagram highlighting the
various flow structures observed in the CFD is provided in Figure 3-13. This axisymmetric representation of the jet flow structure has several regions that are highly dependent on $\rho_L$. Typical to jet flows, the CFD indicates an inviscid jet core, a mixing layer where the inviscid jet and ambient fluid diffuse, and the jet wake which gradually diffuses the energy of the jet. One of the main observations in this work is that with increased values of $\rho_L$, $\theta_{mix}$ increases. This value will be described in detail below, but $\theta_{mix}$ is an indicator of breakdown and instability in the gas-liquid interface. This is consistent with the increased damping behavior of the jet. In addition, the jet wake becomes a bubble mixture. This bubble mixture is quite complex as it involves very large bubbles, breakdown to bubbles, and effects of buoyancy. The new region in this multiphase flow is the intermittent bubble layer region. Previous work has characterized these conditions in the context of a bubble and droplet layer, which is somewhat similar to the present characterization [16]. The intermittent bubble layer is characterized as forming at some length downstream of the nozzle (after the Kelvin-Helmholtz instability develops) and grows at a finite angle.

![Diagram of jet flow structures](image)

**Figure 3-13: Figure indicating regions observed in the multiphase jet interaction**

The parameters in the diagram presented in Figure 3-13 are plotted for all of the cases in Figure 3-14. The inviscid core of the jet is the region in which turbulent mixing is low and the jet is stable, which can be visualized in Figure 3-8. The mixing layer can be similarly visualized in Figure 3-6 and Figure 3-8, and is a mixing region between the jet and ambient fluid. The parameter $\theta_{mix}$ describes the angle from the edge of the inviscid core to the edge of the mixing layer. The intermittent bubble layer is similar to the
mixing layer and is seen in cases with higher values of $\rho_L$. This intermittent bubble layer is the beginning of the interface breakdown and contains shed pockets of gas. The parameter $\theta_{BL}$ describes the angle from vertical (wall parallel) to the edge of this bubble layer, thus a smaller angle indicates a larger spread of the bubble layer. The length of the jet from the wall to the start of the intermittent bubble layer is denoted by $L_{BL}$, while the length of the inviscid core is denoted by $L_{IC}$. The jet wake is the region after jet penetration, where the jet has broken down into primarily a bubble dominated region. In Figure 3-14 (a) and (b), $L_{IC}$ and $L_{BL}$ are shown. The general trend of these two lengths is very similar. Note the crossover behavior observed in $L_{BL}$ due to the over-expanded jet exciting the instability at $\rho_L = 100 \text{ kg/m}^3$. In Figure 3-14 (c) and (d), the angles measured from the CFD are compared. Figure 3-14 (c) indicates that the CFD is predicting that the mixing layer expansion rate for the over-expanded condition (NPR=0.08) is faster than for the under-expanded jet.

![Images of graphs showing length of inviscid core, length of bubble layer initiation, mixing layer angle, and bubble layer angle vs. liquid density.](38)

**Figure 3-14:** Measurements of observed jet regions with variations in $\rho_L$ for the two nozzle conditions.
It is also worth interrogating the effect of $\rho_L$ on the damping behavior in the $M$ and $LPR$ oscillations. This dampening directly correlates to the jet mixing behavior. The oscillations outside of the nozzle are visualized in Figure 3-10 and Figure 3-12 and show that as $\rho_L$ increases, $M$ and $LPR$ experience fewer shocks and a faster return to ambient conditions. This effect is categorized by a damping coefficient analysis, shown in Figure 3-15. Figure 3-15 shows a trend of increased damping with increased $\rho_L$.

Focusing on Figure 3-16, which depicts the amplitude of the first $M$ and $LPR$ oscillations, it can also be observed that as the overall amplitude of the oscillations tends to decrease $\rho_L$ increases. Figure 3-15 and Figure 3-16 quantifies what could be seen in Figure 3-10 and Figure 3-12, that as $\rho_L$ is increased, the overall system experiences a faster breakdown and damping effects on the jet output.
Conclusions

In the above chapter, CFD was used to improve the understanding of the effects of liquid density on submerged gas jets. Similar to previous experimental studies, the CFD model predicted that instabilities develop in the gas-liquid interface as the ambient fluid density increases, causing shorter jets with a wider spread. In addition, the “back-attack” phenomenon from Shi [5,6] was observed, indicating that the ambient fluid and the jet flow backwards and interact with the nozzle.

The main findings are that with increased liquid density, the turbulence level increased dramatically at the gas-liquid interface. This leads to a rapidly developing mixing layer and earlier breakdown of the jet’s shock structure. This results in a shorter penetration length of the gas into the surrounding fluid. This gas-liquid interaction is dominated by the density, with the beginnings of the significant breakdown occurring in the 100 $kg/m^3$ density and above, indicating that this could be a turning point. It was also observed that the nozzle flow type is not the dominating controller of the gas-liquid interactions, though it has some effect on the instability development downstream of the actual nozzle. Other significant observations were the dramatic rise in turbulence, which implies that the phases will mix and diffuse energy as the liquid density increases. The last observation of note is the bulges of gas leaving the jet which appear to transfer momentum to the liquid, as observed large pressure pockets downstream of the jet.
Chapter 4

Gas Jets in a Liquid Co-Flow

Similar to Chapter 3, the following chapter is focused on submerged multiphase jets. However, in order to better focus on the large scale structures, the modeling changes to a 2D axisymmetric model. The cases focus solely on air being jetted into water, employing varying mass-flow rates of the jet, and varying velocities of the liquid flow, which is moving in comparison to the unmoving flow of Chapter 3. The effect of these two changing parameters is investigated with respect to the formations of the submerged jets.

The understanding of multiphase jets, particularly submerged gas jets in a higher density fluid, are important to industrial uses. Chemical and thermal mixing, as well as nuclear reactors all depend on the dynamics of these interactions [4-6]. The more known about the internal and external properties of such systems, the better these systems can be both designed and maintained.

This investigation aims to create a regime map to provide an informed characterization of the different modes formed in these jet cases. The regime map considers five different modes, which occur for different conditions. In addition, the internal flow within these submerged jet formations is evaluated to understand the internal structure of the gas cavities formed.

Methods

The following section details the numerical approach, as well as geometrical modeling, meshing, computational validation, and a computational approach test matrix.
Numerical Methods

The commercial CFD code, Star-CCM+ [11], is utilized for all of the following simulations. A homogeneous, Eulerian-based multiphase-modeling approach is employed, which conserves phase mass through a volume-of-fluid like formulation. Sharp interfaces are maintained through the HRIC scheme [12]. The CFD model is a compressible pressure-based, segregated-flow model that conserves mass, momentum, and energy. The method is not based on conservative form, thus for compressible multiphase flow some modeling errors are expected (and assumed relatively small). The numerical scheme is formally second-order accurate in time and space.

In each case, the jet is modeled as a compressible gas with fluid properties corresponding to air. The outer flow is modeled as water. In terms of the physics of the system, a Reynold’s Averaged Navier-Stokes (RANS) modeling of turbulence is employed that is based on the Spalart-Allmaras (SA) turbulence model. To reduce mesh requirements in the near-wall viscous region resolution, an all-\( \gamma+ \) wall treatment method is used. In terms of temporal resolution, an adaptive time-step approach is used that maintains a temporal resolution with a minimum convective Courant number (throughout the entire flow domain) below unity. Lastly, the present results use an axisymmetric model to reduce simulation time. Because of this, gravity is not considered for the flow-regime map. However, note that gravity is considered in the axial direction for a validation case.

Geometric Configuration

The geometry of the simulation is modeled as a 2D axisymmetric system. The jet is centered in a domain with dimensions of 20 m long by 5 m tall. The nozzle geometry is based off of work by Shi [5] and is designed to be a Mach 2 jet at ideal conditions. The throat has a diameter of 0.0043 m, and the exit has a diameter of 0.0056 m. The length of the entire converging-diverging nozzle is 0.06 m, and the distance between the throat and the nozzle exit is 0.01 m (Figure 4-1). The gas is supplied through a tank
that is 0.02 m in diameter and 0.33 m in length (Figure 4-1). The nozzle itself is located in the center of the domain, 10 m from both the inlet and outlet domain boundary conditions, as well as 5 m from the top of the domain. The jet is also positioned inside of an axisymmetric body that is not faired into the jet. Hence, when the body/jet configuration is in motion, the flow is expected to separate.

**Validation**

To validate the 2D axisymmetric modeling, experiments from Weiland are used [4]. In these experiments, a converging-diverging nozzle is used to jet vertically into a tank of still water. The chamber has a width and depth of 0.46m, with wave breakers located 0.46m above the jet nozzle. The nozzle is rapid prototyped with constant exit area and a changing throat area, in order to produce perfectly expanded jets with varying Mach values. The computational domain and conditions approximate those from the experiments, testing several different Mach jets within a vertical chamber. In this case, gravity is applied to act against the jet (note the remaining cases ignore gravity). Note that the nozzle geometry is specific to the validation, and the subsequent results use the nozzle reported in Shi et al. [5]. The jet condition achieved in the CFD model was quite close to the experiment, where a jet Mach number in the CFD was 0.86 as compared to the value of 0.91 as measured in the experiment. An overview of
comparisons to the experiments for different Mach numbers is provided in Table 4-1 and Table 4-2. A sample qualitative comparison of the gas content measured from the experiment (Figure 4-2 (a)) can be compared to CFD results (Figure 4-2 (b)). For Figure 4-2 and, Figure 4-3 the experimental data is a time average of gas location from the experiment, taken from photos where the visible location of the gas spread is extracted, while the simulation data is a time average of the volume fraction. Hence, a time averaged volume fraction is only a qualitative validation metric.

As observed in Figure 4-3, where they are co-plotted, the predicted volume fraction shape correlates well to the experiment. Note that in these plots the spatial scale is normalized by $L_Q$, which is the square root of the nozzle exit area. Although the core volume length is not exactly matched, the exit shape of the jet is. In terms of these comparisons, there are several important clarifications:

- The nozzle exit diameter is not explicitly stated in either Weiland’s dissertation or journal paper [4,7], however, the diameter can be estimated from figures, and the fact that 0.8 mm is equivalent to $y/L_Q = 0.14$. $L_Q$ is given by Weiland as the square root of the nozzle exit area. It is reported that the exit diameter is held constant and that the throat is changed to create nozzles designed for different Mach numbers. The throats are calculated using quasi-steady area ratios based on tabulations from *Modern Compressible Flow* by Anderson [17]. This involved the use of a table of isentropic flow properties to calculate the necessary nozzle throat area to produce a perfectly expanded jet for given Mach and exit area properties.

- As the primary focus of this effort is in the near-jet region and character, the reader should focus on the visual comparison of the near-jet region. In that region, there is excellent correlation of the CFD with experiment in terms of the core length and jet shape.

- The long range spread rate appears to display more deviation, but such a lack of agreement is not a surprise. First, the discrepancy in the measurements method and the CFD is expected to be amplified in the lower volume fraction regions. This is due to the CFD under predicting this spread, while the spread may be amplified by stray bubbles in the experimental images. In
addition, it would likely require a model that explicitly models slip in the bubble and liquid motion (which is not considered in a homogenous model).

Table 4-1: Comparison of Jet Characteristics from Weiland Experiment [4] and CFD

<table>
<thead>
<tr>
<th>Case</th>
<th>Source</th>
<th>Jet Total Pressure $P_o$ (Pax10$^5$)</th>
<th>Exit Mach $V_J/a$</th>
<th>Exit Pressure, $P_e$ (Pax10$^5$)</th>
<th>Ratio of Exit to Freestream, $P_e/P_H$</th>
<th>Ratio of Exit to Stagnation, $P_e/P_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Weiland [4]</td>
<td>1.37</td>
<td>0.61</td>
<td>1.06</td>
<td>1.02</td>
<td>0.78</td>
</tr>
<tr>
<td></td>
<td>Simulation</td>
<td>1.38</td>
<td>0.59</td>
<td>1.07</td>
<td>1.01</td>
<td>0.77</td>
</tr>
<tr>
<td>2</td>
<td>Weiland [4]</td>
<td>1.61</td>
<td>0.77</td>
<td>1.09</td>
<td>1.05</td>
<td>0.67</td>
</tr>
<tr>
<td></td>
<td>Simulation</td>
<td>1.61</td>
<td>0.71</td>
<td>1.13</td>
<td>1.07</td>
<td>0.70</td>
</tr>
<tr>
<td>3</td>
<td>Weiland [4]</td>
<td>1.96</td>
<td>0.91</td>
<td>1.15</td>
<td>1.10</td>
<td>0.58</td>
</tr>
<tr>
<td></td>
<td>Simulation</td>
<td>1.97</td>
<td>0.86</td>
<td>1.20</td>
<td>1.08</td>
<td>0.61</td>
</tr>
</tbody>
</table>

Table 4-2: Percent Differences Between Experiment [4] and Simulation

<table>
<thead>
<tr>
<th>Case</th>
<th>Mach</th>
<th>$P_e/P_H$</th>
<th>$P_e/P_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.28%</td>
<td>0.98%</td>
<td>1.28%</td>
</tr>
<tr>
<td>2</td>
<td>7.79%</td>
<td>1.90%</td>
<td>4.48%</td>
</tr>
<tr>
<td>3</td>
<td>5.49%</td>
<td>1.82%</td>
<td>5.17%</td>
</tr>
</tbody>
</table>

![Figure 4-2: Time Averaged Gas Location of Experiment and Mean Gas Volume Fraction of the CFD](image)
Computational Mesh

The simulation is meshed as an 2D axisymmetric flow. A quadrilateral-based mesh approach is employed with thin prism layers to resolve viscous layers and help maintain sharp corners and curves. There is also a refinement region in the near-gas jet region, which is used to refine the jet-liquid interaction region (Figure 4-4). A single parameter was used to govern the mesh resolution. In order to accelerate the remove of transient solutions, this parameter is decreased in a sequential process that reduced the relative mesh size from 5, to 3, to 1. In addition to reducing the time required to remove the transient part of the solution, the process also provides insight into the mesh sensitivity of the results. In general, there was not much mesh sensitivity observed between the 3 and 1 mesh levels.
Matrix of Conditions Explained

A test matrix of conditions are used to evaluate and characterize the behavior of the gas jet for many conditions. Several parameters are varied that include the liquid flow velocity and the mass flow rate of the jet. The jet mass flow rate and liquid velocity were simulated for every combination of the conditions: $V_\infty$ of 1, 5, 10, 15, 25, 30, 40, and 50 m/s, $m_{jet}$ of 0.0025, 0.0040, 0.0050, 0.0060, 0.0075, 0.0080, 0.0100, and 0.0125 kg/s. This produced 64 total cases. As the coarse mesh captures the large scale features, it is the focus of the following discussions. This allowed the largest variation of $V_\infty$ and $m_{jet}$ in a reasonable time frame with good simulation stability.
Results

Parameter Effects

Figure 4-5: Color contours indicate Mach number. Contour lines indicate the gas/liquid interface. The x-axis varies the jet mass flow. The y-axis varies the liquid velocity. These shots are all at the end of the simulation time. The images have been compressed slightly in the X-axis.
Both liquid velocity and jet mass flow rate are parameterized to develop a global understanding of the behavior of the gas-jet/liquid-flow interaction. For this purpose, the cases were compiled into a matrix of images that includes the liquid velocity, on the ‘Y-Axis,’ and the gas mass flow rate, on the ‘X-Axis.’ This compiled image matrix contains two scalars, where the color contours indicate Mach (note that the scale is the same for every image) and the grayscale lines indicating gaseous volume fraction (with values within white being all gas, and values outside the black lines being all liquid). In general, in Figure 4-5, the jet-liquid interaction clearly displays different characteristics throughout the conditions evaluated. These various cavity modes are what the present investigation aims to characterize.

**Effect of Freestream Velocity \( (V_\infty) \)**

In the context of Figure 4-5, first consider the impact of the freestream velocity. The effect of an increasing free-stream velocity can be observed by moving vertically upwards at any given jet-mass flow rate. At the lowest speed (1 m/s) all jet flows display a interaction between the gas and liquid. This jet has an unstable interface that sheds large bubbles and appears to pulsate. As the external velocity is increased to 10 m/s, the jet gas attaches to the aft part of the pseudo body and forms a gas-filled cavity. At the lowest mass flow rate (0.0025 kg/s) the cavity formed is similar to the top view of a toroidal cavity from [1] (Figure 4-11). As the mass flow rate is increased, the cavity evolves to an intermediate cavity that ejects pockets of gas, before transitioning to a larger stable cavity. As the external velocity is further increased, the overall cavities become larger but also more unstable. With increasing freestream velocity, the gas-liquid interface becomes more unstable, with faster fluctuations (this is supported by the corresponding videos to each case) These observations are further supported by full videos of the simulations in addition to the single snapshot of each case taken at the end (2s for the coarse cases).
Effect of Mass Flow Rate of the Jet ($\dot{m}_{\text{jet}}$)

Now consider the impact of increasing the mass flow rate of the jet, which can be observed by moving horizontally, to the right, in Figure 4-5. In this process, an increase in $\dot{m}_{\text{jet}}$ changes both the ventilation rate of the gas-filled cavity (affecting the overall cavity dynamics), as well as the exit conditions of the jet (changing the flow features and pressure waves surrounding the jet exit). The general observation is that at a higher mass flow rate, a larger, more stable cavity forms. In addition, there is a clear impact of free-stream velocity, which tends to require more gas to maintain the same cavity type.

Discussion

Within the results section, the effect of the fluid velocity and mass-flow rate of the jet are examined. These parameters pertain to the resulting character of the jet-liquid interaction. The images show that when the fluid velocity is low, an oscillating jet structure develops. As the fluid velocity is increased, the jet begins to form ventilated cavities. The character and stability (or steadiness) of these cavities appears to be affected by both the fluid velocity ($V_\infty$) and mass-flow rate of the jet ($\dot{m}_{\text{jet}}$). In the following sections, a global understanding of these modes is developed, and the jet-liquid modes are classified into regimes. In addition, the effect of these interactions on the flow structures is investigated.

Jet Flow Regime Map

Through investigation of the case matrix (Figure 4-5), five regimes have been identified. Note that these are more apparent in the video of each simulation. These five regimes are defined as follows:

1. Shedding Jet - Large bubble ejections, attached to the nozzle exit
2. Toroidal Cavity - Small gaseous cavity, attached to the jet casing. Constant stream of liquid entering the cavity


5. Over-Ventilated Cavity – Small gaseous cavity attached to the jet casing. Large bubble ejections through the rear of the cavity.

Regime Map Overview

Figure 4-6 shows an overview of the regime map based off general trends and the observed regimes. This map outlines the observed gas-jet liquid interaction, displaying the complexity of the interaction in the present configuration. Here we see Regime 1 (shedding jet) for at the low values of $V_\infty$. This regime corresponds to gas jets in stagnant flow such as discussed by refs [4-7]. As $V_\infty$ increases, so does the complexity of the gas-liquid interaction. First of all, a gaseous cavity forms and attaches to the pseudo body. For low $\dot{m}_{jet}$ values, the toroidal cavity forms (Regime 2). As $\dot{m}_{jet}$ increases, the cavity grows and begins to pulsate (Regime 3). As $\dot{m}_{jet}$ is further increased, the cavity transitions to a stable cavity (Regime 4), and eventually an overventilated cavity (Regime 5). The $\dot{m}_{jet}$ required to transition to each of these regimes is clearly a function of $V_\infty$. Each of these regimes are further discussed below while refinement of the lines delineating the Regimes in Figure 4-6 is developed in Chapter 5.
Regime 1: Shedding Jet

For a low freestream velocity (i.e., 1 m/s), the “Shedding Jet” regime was observed for all jet conditions. This regime is present in the bottom row of Figure 4-5. When in this regime, the center of the jet (internal to the gaseous bubbles) interacts with the bubble in a pinching and expanding manner, (visible in Figure 4-7). The interconnected bubbles caused by the expansions then travel downstream. By observing the time average of the case (Figure 4-8), the nozzle exit condition can be seen to form a consistent pinch point, followed by a downstream spreading of the gas away from the nozzle, with consistently sized bubbles. Unique to this condition (as apparent in Figure 4-5), the gas does not attach to aft part of the body, but rather attaches in the region of the jet exit (as was observed in the validation case). It is apparent that, as the gas mass flow rate increase, the severity of the interface instability also increases. Hence, the interface instabilities correlate to the energy in the jet mass flow. In addition, with
increased gas-mass flow, these large-scale pulsations and shedding events become sharper and more erratic.

Flow within the Shedding Jet regime is relatively straightforward, with flow velocity and direction affected by the gas-liquid interface, but all traveling in generally the same direction (Figure 4-9, Figure 4-10). By observing the streamlines in Figure 4-9 (a), it can be seen that the streamlines travel through the center of the series of bubbles, with a constriction at the jet exit. The vector scene (Figure 4-9 (b)) supports this central movement and the constriction on the jet, as the flow coming around the jet casing and constricting the gas flow at the nozzle exit. By observing the time averaged flow of the same case (Figure 4-10), it can be seen that the majority of the gas velocity is contained within the centerline of the shedding jet, and by observing the vectors in Figure 4-10 (b) it can be seen that over time the jet exit is constricted by the flow from around the jet casing, despite the occasional bubble expansions witnessed in Figure 4-7 at about 1.4s and 1.8s.
Figure 4-7: $V_\infty = 1$ m/s, $m_{\text{jet}} = 0.0050$ kg/s, Time Sequence Figure
Figure 4-8: \( V_\infty = 1 \text{ m/s}, \ m_{\text{jet}} = 0.0050 \text{ kg/s}, \) Time Averaged

(a) Streamlines from the Jet Chamber

(b) Vector Scene of All Flow Velocity

Figure 4-9: \( V_\infty = 1 \text{ m/s}, \ m_{\text{jet}} = 0.0050 \text{ kg/s}, \) Snapshot
Regime 2: Toroidal-Cavity Formation

As apparent in the upper rows of Figure 4-5, as exterior liquid velocity increases, the gaseous bubble attaches to the jet casing and forms a cavity in the wake of the casing. The resulting cavity forms different regimes. For a low jet mass-flow rate, this cavity has a structure like a toroidal-vortex cavity.
From understanding developed in supercavitation, this recirculating type cavity has a liquid-mixture that periodically flows back towards the jet exit and is observed in the present work as highlighted in the sequences of images in Figure 4-11. This sequence indicates an oscillatory nature of the attached cavity, with the top and bottom ‘flaps’ containing a large wave traveling along the interface. The sequence also shows the steady stream of ejected clouds of gas from the end of the cavity. These ejected clouds are also clearly visible in Figure 4-12, which is the time-averaged representation of the flow. Interestingly, near the jet there is a pocket of air directly coming from the jet that appears to be competing with an upstream moving, impinging jet of liquid, which is particularly evident at about 1.6s and 1.8s, in Figure 4-11. The overall flow structure indicates strong recirculation throughout the cavity and that large bubbles shed from the tail of the cavity.

The interaction of the gas jet with the reentrant liquid is further supported by Figure 4-13(a), a snapshot of the jet containing streamlines leaving the jet nozzle and being forced backwards by the interaction with the liquid. The same is true of the time averaged streamlines, visible in Figure 4-14 (a). The time average reveals that this is a consistent phenomenon through the jet-cavity’s history, with an average reentrant length meeting an average jet penetration. Both streamline images also reveal that the gas leaving the jet is typically pushed back to the jet casing, where it is then entrained into the interface before exiting the cavity. The close up vector views of the cavity shown in Figure 4-13 (b) and Figure 4-14 (b), the single time step and the time average, respectively, show recirculation within the fully gaseous regions of the cavity. This is indicated by the flow moving towards the jet in the center of the cavity, but away from the jet along the cavity interface. Particularly in the time average image, it can be seen that this recirculation is a consistent feature of this jet, with flow moving clockwise in the upper portion of the jet cavity, and counter clockwise on the lower portion.

Overall, this regime appears to form for the lowest gas mass-flow rates. In terms of the mass flow rate necessary to support this regime, as well as the stability of the interface, this regime occurs as the transition between the shedding jet and the small pulsating cavity.
Figure 4-11: $V_\infty = 10$ m/s, $m_{jet} = 0.0025$ kg/s, Time Sequences
Figure 4-12: \( V_\infty = 10 \text{ m/s}, m_{\text{jet}} = 0.0025 \text{ kg/s}, \text{Time Average} \)

(a) Streamlines from the Jet Chamber

(b) Vector Scene of All Flow Velocity

Figure 4-13: \( V_\infty = 10 \text{ m/s}, m_{\text{jet}} = 0.0025 \text{ kg/s}, \text{Snapshot} \)
Regime 3: Pulsating Cavity

As the jet-mass flow increases, the pulsating-type cavity forms. This regime is more stable than that of the toroidal-cavity, however, it has not yet fully stabilized. In this regime, the cavity takes the form of a small stable cavity that changes in length as it grows and then ejects pockets of air. An example is
presented Figure 4-15, which provides an example result at various points in time. In the plot, it can be observed that at the end of the cavity, small pockets of liquid reenter the cavity at the aft end of the cavity (much like a reentrant jet). A time-averaged plot is provided in Figure 4-16, where the reentrant jet becomes more apparent in the gas/liquid mixture along the centerline. Unlike the toroidal cavity, the cavity is fully formed in the core and does not present with the same ‘claw’ shape as the toroidal. Hence, the water entering the cavity is not strongly “attacking” the jet exit. Lastly, the pulsation character of the cavity can be observed in either the waves forming on the cavity in Figure 4-15 and through the large spatial variation in the volume fraction in the time-averaged plot (Figure 4-16).

Similar to the Toroidal-Cavity, the interaction between the jet and liquid entering the cavity can be seen in the streamlines. Figure 4-17 (a) captures a moment in which a large pocket of liquid is pressing on the jet, causing the penetration depth to be truncated. The time average of this jet cavity, visible in Figure 4-18 (a) shows a constant stream of liquid reentering the cavity, though contours are lighter than those in the Toroidal-Cavity, indicating that the volume fraction of liquid constantly entering the cavity is lower. Additionally, the time average of the Pulsating Cavity does not have well-defined prongs at the end like the Toroidal-Cavity, indicating that the end of the cavity is relatively closed, without a large continuous stream of liquid entering the cavity. The time averaged streamlines of the Pulsating Cavity also show that over time the flow from the jet is being recirculated through the cavity. The recirculation is further supported in the vector displays in Figure 4-17 (b) and Figure 4-18 (b), where close-ups show the internal flow being redirected in the corners between the jet casing and the interface. The vector plots also show that the majority of the internal flow is moving towards the jet, with the flow away from the jet being dominated by the interface and areas very close to the interface. This indicates that the majority of the gas exits the system along the interface as small pockets of gas.

Overall, the Pulsating Cavity regime seems to be defined by mostly gaseous core with small pockets of liquid entering the system. The gas-liquid interface presents with waves, though they are smaller than the Toroidal-Cavity regime. Gas is ejected from the cavity as small bubbles from the cavity
closure. This regime requires more mass-flow rate than the Toroidal-Cavity, but less than the Stable Cavity.

Figure 4-15: $V_{\infty} = 25 \text{ m/s}$, $m_{jet} = 0.0050 \text{ kg/s}$, Time Sequence
Figure 4-16: $V_\infty = 25 \text{ m/s}$, $m_{\text{jet}} = 0.0050 \text{ kg/s}$, Time Average

(a) Streamlines from the Jet Chamber

(b) Vector Scene of All Flow Velocity

Figure 4-17: $V_\infty = 25 \text{ m/s}$, $m_{\text{jet}} = 0.0050 \text{ kg/s}$, Snapshot
Regime 4: Stable Cavity

As the gas flow rate further increases, the cavity stabilizes forming the stable-cavity regime. In this regime, a large stable gaseous cavity attaches to the jet casing. This formation appears to form by reducing reentrant pockets of liquid that were characteristic to the Toroidal and Pulsating cavities. In this
regime, the most stable, largest cavity is observed. This stability is indicated by the reduction of liquid entering the cavity to almost none, as well as the smaller fluctuations on the gas-liquid interface when compared to the other regimes. The time sequence of the cavity, shown in Figure 4-19, shows that over the last second of the simulations time that no more liquid is reentering the cavity. The sequence also shows that the interface is virtually unchanged between each snapshot, further supporting the fact that the interface fluctuations are very small in this regime. The time average of the case, Figure 4-20, indicates that some liquid reentered the cavity over its earlier formation, but generally the entire cavity is made up of all gas. The time average also further supports the stability of the interface, as there is little smearing or blurring of the grayscale contour lines making up the interface. This shows that the interface is roughly always the same width, supporting small fluctuations, rather than large interfacial waves.

Looking at the streamlines for a single time step, Figure 4-21 (a), it can be seen that the jet is able to penetrate much further into the cavity before being repelled, compared to both the Toroidal and Pulsating cavities. However, in this case, the jet’s maximum reach does not seem to be controlled by interaction with reentrant liquid, but rather reacting with the recirculating flow downstream in the cavity. The time average of these streamlines (Figure 4-22 (a)) are very similar, further supporting that the form of the Stable cavity is indeed the most stable regime, with little change occurring with respect to time. The vector display of the snapshot (Figure 4-21 (b)) shows the flow near the nozzle, and the way that the gas exiting the nozzle interacts with the gas-liquid interface of the cavity. The recirculation of the main cavity is indicated by the flow towards the nozzle from the aft of the cavity, but there is also corner recirculation between the jet casing and the interface. This shows that the jet begins to act like a confined jet near the nozzle, entraining its own gas back into the system through the corner. The time average (Figure 4-22 (b)) shows the same phenomenon. Both the vectors and streamlines indicate that much of the gas leaving the system travels along the interface, as through the center of the cavity there is flow heading towards the jet.
Overall, this regime is classified as the Stable cavity for good reason. There is little to no liquid reentering the system, and the interface has few instabilities. This regime requires a large mass flow rate, and is easily destabilized by increasing freestream velocity.

Figure 4-19: $V_\infty = 10 \text{ m/s}$, $m_{\text{jet}} = 0.0100 \text{ kg/s}$, Time Sequence
Figure 4-20: $V_\infty = 10 \text{ m/s}, m_{\text{jet}} = 0.0100 \text{ kg/s}$, Time Average

(a) Streamlines from the Jet Chamber

(b) Vector Scene of All Flow Velocity

Figure 4-21: $V_\infty = 10 \text{ m/s}, m_{\text{jet}} = 0.0100 \text{ kg/s}$, Snapshot
Regime 5: Over-Ventilated Cavity

The last regime observed in this effort only appears once in the set of cases evaluated ($V_\infty = 5$ m/s and $\dot{m}_{\text{jet}} = 0.0125$ kg/s). This regime appears to occur for high values of $\dot{m}_{\text{jet}}$, and takes place when the stable cavity regime can no longer sustain the amount of gas being injected into the cavity, leading to an
over-ventilated condition. This condition is characterized by the small cavity with no reentrant liquid, which expels large pockets of gas from the rear of the cavity. These large pockets are expelled in a similar manner to the Shedding Jet, and this regime is very much like cavity with a jet in the exit. This formation can be observed in the time sequence provided in Figure 4-23. This sequence shows a small expanding and contracting cavity attached to the jet casing, out of which a constant stream of gaseous bubble are expelled. The time-averaged representation, from Figure 4-24, indicates a smaller cavity with a wake composed of a mixture of gas and liquid, in which the wake portion is very reminiscent of Figure 4-8, the time average of the shedding jet cavity.

A snapshot of the streamlines supports the idea that this is a combination of a cavity and a shedding jet (Figure 4-25 (a)). Near the nozzle, within the cavity portion of the system, the jet presents with the same steady streamlines, similar to Figure 4-21 (a), the jet within the Stable Cavity. However, rather than continuing to exist within a gaseous cavity, the cavity begins to break apart, after which the streamlines spread like Figure 4-9 (a), the Shedding Jet regimes. The time averaged streamlines (Figure 4-26 (a)) show this same transition from a gaseous contained jet, to the spreading through serious of bubbles, or a bubbly gas-liquid mixture. The vector images (Figure 4-25 (b) and Figure 4-26 (b)) show the same combination of traits from the Stable Cavity and Shedding Jet retimes. Near the jet, the flow can be seen to consist of corner recirculation, as in the Stable Cavity case. However, when the cavity reaches its break-up or pinch-off length, the flow begins to spread and continue away from the jet. This is unlike any of the cavity regimes, which all have cavity recirculation and flow moving towards the jet along the centerline. Rather, this nature that the flow continues downstream, away from the jet, is consistent with the flow behavior of the Shedding Jet regime. This is consistent with the visual analysis of the volume-fraction isolines, which show that the gas breaks up and travels downstream in bubbles (snapshots), or that the time average of the volume fraction shows a large jet spreading.

This regime requires the most mass-flow rate, likely due to its over-ventilated nature. In terms of transitioning from one regime to another, this appears to occur after the Stable Cavity. This means that if the freestream is held constant and the mass flow increased, that the cavity formations will transition from
Toroidal, to Pulsating, to Stable, and finally to Over-Ventilated, confirming that this is the highest mass flow regime witnessed in this study.

Figure 4-23: $V_\infty = 5 \text{ m/s}, \dot{m}_{\text{jet}} = 0.0125 \text{ kg/s}$, Time Sequence
Figure 4-24: $V_\infty = 5$ m/s, $m_{jet} = 0.0125$ kg/s, Time Average

(a) Streamlines from the Jet Chamber

(b) Vector Scene of All Flow Velocity

Figure 4-25: $V_\infty = 5$ m/s, $m_{jet} = 0.0125$ kg/s, Snapshot
Conclusions

In this chapter, a 2D axisymmetric model for a submerged multiphase jet was validated for a range of conditions. The effort then investigated the modes of the interactions of the gas jet with the
surrounding liquid, while varying the parameters $V_\infty$ and $m_{\text{jet}}$. The findings were analyzed for the effect of each parameter varied. This lead to the development of a preliminary regime map for these interactions (similar to those used in pipe flow). The five regimes classified (Regime 1, Shedding Jet; Regime 2, Toroidal-Cavity; Regime 3, Pulsating Cavity Regime 4, Stable Cavity; and Regime 5, Over-Ventilated Cavity) can be characterized with liquid velocity and the jet’s mass flow rate. The effect of the varying large scale structures present in each regime were evaluated in terms of the internal flow of the jet-cavity formations. The gas-liquid interactions cause expansions and contractions of the jet flow right out of the nozzle, more so in the lower numbered regimes (such as the Shedding Jet and Toroidal Cavity). The worry with this behavior is in chemical mixing purposes of jet use, where it can cause the reactions to take place on the jet and surrounding equipment, reducing the lifespan of such equipment. Similar behavior has been called attention to in the form of “back-attack” by Shi [6]. These findings indicate that operating within specific regimes may increase the lifespan of industrial equipment, however an explicit study of how these regimes related to chemical mixing has not yet been conducted.

This chapter provides a good foundational analysis of being able to classify these jet regimes. However, the effects of things such as the compressibility of the gas and liquid, as well as nozzle geometry and other parameters (such as gas properties) are not yet clear. Thus, it will be important to include more controlling parameters to the analysis. In addition, using this data to create usable regime maps is of great interest and will be investigated in the next chapter, along with several non-dimensional parameters.
Chapter 5

Regime Map Development for Gas Jets in a Liquid Co-Flow

The following chapter uses the simulations and results from Chapter 4 to develop a regime map of the parametric studies. The regimes discussed in Chapter 4 are each assigned a numeric value, which is used to help plot the regime map. Assigning this value is based on studying each case in terms of an animation and as a time average. Using the numeric regime values, each specific case is numbered, with transitioning cases assigned a half-value. The map is assessed with both the varied parameters ($V_\infty$ and $m_{\text{jet}}$) as well as some non-dimensional parameters. The structure of the resultant regime maps is discussed and conclusions are made.

Methods

Regime Classification of Cases

The first step in the process involves assigning a numeric value to each flow regime occurs. Assigning a regime to each simulation is based on visual assessment of both video from the unsteady simulations and the time-averaged solution. In this visual assessment, the flow structure associated with each regime (discussed in Chapter 4) is used to determine the regime. Based on this visual assessment, each case is assigned a numeric value that corresponds to a specific regime. Recall that, as classified in Chapter 4, these regimes are: Shedding Jet, Toroidal Cavity, Pulsating Cavity, Stable Cavity, and Over-Ventilated Cavity. As these flow regimes are identified, each condition is assigned a numeric value based on the values provided in Table 5-1. Examples of each of these regimes, using images from the time-averaged data, are given provided in Figure 5-1. In these images, distinct characteristics can be observed that are associated with each regimes given by: (2) the toroidal vortex (and it strong reentrant jet) in Figure 5-1 (a), (3) the weak reentrant jet of the pulsating cavity in Figure 5-1 (b), (4) the large stable
cavity in Figure 5-1 (c), (5) the smaller cavity with an inflection point associated with the over-ventilated cavity in Figure 5-1 (d), and (6) the diffused, unattached cavity provided in Figure 5-1 (e). The data are then tabulated to develop flow-regime maps based on the numerical assignments. For ease of the later linear interpolations between numeric regimes, the Shedding Jet regime has been given the value of 6 rather than 1. This decision should become apparent looking at the regime map visuals below, but the assigned numeric values for all further regime mapping plots are given below.

<table>
<thead>
<tr>
<th>Table 5-1: Table of Numeric Regime Assignments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Toroidal Cavity</td>
</tr>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

Figure 5-1: Time-Average Example of Each Regime

Results

As previously mentioned, the development of the flow regime map utilizes the time-averaged results. The full spread of time-averaged simulation images are provided in Figure 5-2. This includes the results for each jet-gas mass flow rate (which increases on the x-axis) and liquid flow velocity (increasing on the y-axis). Once each CFD simulation is given a numeric regime classification, the result is tabulated and the overall results are provided in Table 5-2. As can be seen, some of these cases have been assigned
half-values. These result from ambiguities in determining the flow regime, which are not always clearly distinguished. Hence, cases are in such transitional regimes are assigned half-values between the two regimes. With these results, a flow regime flow-regime map can be determined.

Figure 5-2: Time-Averaged Matrices of Simulation Cases for Regime Classification (Images have been stretched in the y-axis)
Table 5-2: Table of Case Regime Assignments

<table>
<thead>
<tr>
<th></th>
<th>0.0025 kg/s</th>
<th>0.0040 kg/s</th>
<th>0.0050 kg/s</th>
<th>0.0060 kg/s</th>
<th>0.0075 kg/s</th>
<th>0.0080 kg/s</th>
<th>0.0100 kg/s</th>
<th>0.0125 kg/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>50 m/s</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>2.5</td>
<td>2.5</td>
<td>2.5</td>
<td>2.5</td>
<td>3</td>
</tr>
<tr>
<td>40 m/s</td>
<td>2</td>
<td>2</td>
<td>2.5</td>
<td>2.5</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>3.5</td>
</tr>
<tr>
<td>30 m/s</td>
<td>2</td>
<td>2.5</td>
<td>2.5</td>
<td>2.5</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>25 m/s</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>2.5</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>3.5</td>
</tr>
<tr>
<td>15 m/s</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>3</td>
<td>3.5</td>
<td>3.5</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>10 m/s</td>
<td>2</td>
<td>3</td>
<td>3.5</td>
<td>3.5</td>
<td>4</td>
<td>3.5</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>05 m/s</td>
<td>2.5</td>
<td>3.5</td>
<td>4</td>
<td>4</td>
<td>4</td>
<td>4</td>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td>01 m/s</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
</tbody>
</table>

Initiation of the flow-regime map is established by plotting each case and its corresponding numeric value for each $V_\infty$ and $\dot{m}_{jet}$. The resultant plot is displayed below in Figure 5-3, where the x/y-locations correspond to the flow condition, and the color of the symbol corresponds to the flow-regime. In general, the present plot highlight the flow regimes, but does not visually indicate the trends that are expected from a flow-regime map. For now, visual observation shows that the Shedding Jet regime appears only in values of low $V_\infty$, and that the data points transition from larger regime numbers (5, Over-Ventilated Cavities) in the lower right-hand corner (low $V_\infty$, high $\dot{m}_{jet}$) to smaller regime numbers (2, Toroidal Cavity) in the upper left-hand corner (high $V_\infty$, low $\dot{m}_{jet}$). These observations will be used to guide flow regime map development.
Discussion

Flow-Regime Map Development

The flow-regime map is developed based on the refining the results plotted in Figure 5-3. The refinement process involves exploring different methods of regime map creation. The first approach used involves plotting filled contours where each contour color indicates a flow regime. Such an approach requires more data than generated in the CFD, hence, interpolation and various interpolation methods are explored. These are then used to guide the regime-map development.

The first interpolation method explored applies a linear interpolation of the points over the 2D plane created by the various conditions that span $V_\infty$ and $m_{\text{jet}}$. Performing this interpolation, over an equally spaced 16x16 mesh in Matlab, results in Figure 5-4. This method works by creating an equally spaced grid with 16x16 points. Then, Matlab linearly interpolates between the given input data (Figure 5-3) and the mesh points. In this plot, the contour lines are plotted such that they delineate the regimes. In this plot, it becomes more visually clear that the regimes delineations are mostly constructed by diagonal “fans” across the 2D plane. When these interpolated regions are filled, as done in Figure 5-5, it can be observed that several of the CFD results are placed in the wrong regions. This is indicated by a mismatch in the filled contour and the color of the symbol representing the CFD result. Such a mismatch tends to occur most for Stable Cavity regime. In addition, the linear interpolation includes areas for the intermediate regime between Shedding Jet (6, Dark Red) and Over-Ventilated Cavity (5, Orange), as well as the intermediate regime between Stable Cavity(4, Yellow) and Over-Ventilated Cavity (5, Orange). The inclusion of these intermediate regions into the interpolated maps results in many data points landing in regimes inconsistent with the map. Hence, many of the Stable Cavity data points (4, Yellow) are predicted to occur in the pulsating regime. Alleviation of these issues is required to determine an adequate flow-regime map.
The next step involved exploring cubic interpolation to the construction the flow-regime map. Results from the cubic interpolation are plotted in a similar form to the linear-interpolation in Figure 5-6 and Figure 5-7. In Figure 5-6, it is apparent that the lines that seem to better separate the given regime data. However, when the contours are filled as can be observed in Figure 5-7, it can be seen that isolated ‘pockets’ of a regime tend to occur. In addition, many of the Stable Cavity cases are still placed within the surrounding intermediate regime (3.5, Green). From these results, it is not clear that regime maps can be constructed by simply plotting the data and, hence, requires interpretation.

Due to previous difficulties, the regime map is visually created taking the two interpolation approaches into consideration. This map (Figure 5-8) does not follow a strict interpolation method between the given data points, but rather attempts to fill the map in the smoothest method possible. In this plot, the data are correctly positioned into their corresponding numeric regimes or intermediate regimes, with some occurring on the borders. In additions, there are few sharp transitions and no resultant ‘pocket’ regions. For the sake of this analysis, more intermediate data points are necessary for interpolation methods to produce a well-formed regime map, and thus, the hand-divided map will be smoothest result.
Figure 5-4: 16x16 Linear Interpolation of Table 5-2 Data

Over-Ventilated Cavity
Stable Cavity
Pulsating Cavity
Toroidal Cavity
Shedding Jet

Figure 5-5: Filled 16x16 Linear Interpolation of Table 5-2 Data
Figure 5-6: 16x16 Cubic Interpolation of Table 5-2 Data

Figure 5-7: Filled 16x16 Cubic Interpolation of Table 5-2 Data
Non-Dimensional Parameters

In addition to creating a regime map based from the dimensional parameters from the simulation, two sets of non-dimensional parameters were also investigated. The first set is given by equations 5.1 and 5.2. All necessary values were measured from the simulation, using both a probe located upstream of the jet/body for freestream conditions, as well as a probe at the nozzle exit to measure \( V_j \) and \( \rho_j \). The density of the gas, \( \rho_g \), was measured as an average of the density across the cavity (similar to methods utilized in Chapter 2 for isolating the gas cavity). The method for calculating \( V_g \) is given by equation 5.3, with \( D_B \) being the diameter of the pseudo-body around the jet, equal to 0.2m. When the data are plotted to the first set of non-dimensional parameters (Figure 5-9), the result is a spread of points from the lower left-hand corner to the upper right-hand corner, with the majority of data points being clumped together. The spread
also goes from the highest regime values (Shedding Jet, 6) to the lowest regime values (Toroidal Cavity, 2).

\[
\Pi_2 = \frac{\left(\frac{1}{2} \rho_\infty V_\infty^2\right)}{\left(\frac{1}{2} \rho_j V_j^2\right)}
\]

\[
\Pi_3 = \frac{\left(\frac{1}{2} \rho_\infty V_\infty^2\right)}{\left(\frac{1}{2} \rho_g V_g^2\right)}
\]

\[
V_g = \frac{\dot{m}_j}{\rho_g \pi \frac{D_B^3}{4}}
\]

In order to better visualize the spread of data, the same plot is shown on log-log axes (Figure 5-10). The above description of how the data transitions (left to right, high to low) is made clear on the log-line. These non-dimensional parameters create a clear distinction between the Shedding Jet regime (6) and the lower numeric regimes (Stable Cavity, Pulsating Cavity, Toroidal Cavity), but the lower numeric regimes blend together.

\[\text{Figure 5-9: Data Plotted with Equation 5.1 vs 5.2}\]
The second set of non-dimensional parameters are given by equations 5.4 and 5.5. The drag is calculated within StarCCM+ and is the drag on the pseudo-body surrounding the jet. All other values are either measured or given parameters. The corresponding data points are plotted in Figure 5-11. As with the first set of non-dimensional parameters, many of the data points become compressed to a single region. However, the data now goes from lower regime numbers to larger regime numbers as the line goes from the lower left-hand corner to the upper right-hand corner. In addition, the data collapses to a curve rather than a spread.

\[
\Pi_4 = \frac{\dot{m}_j}{\rho_\infty V_\infty \pi \frac{D_B^2}{4}} \\
\Pi_5 = \frac{\dot{m}_j V_j}{Drag} 
\]

5.4

5.5
Figure 5-11: Data Plotted with Equation 5.4 vs 5.5

Figure 5-12: Data Plotted with Equation 5.4 vs 5.5, Log-Log
Again, in order to better separate the data points, the data are plotted on a log-log plot and presented in Figure 5-12. As with the first set of non-dimensional parameters, it can be seen in the log-log plot that only the Shedding Jet regime is clearly separated from the data set. The rest of the regimes tend to blend together. Despite these drawbacks, both sets of non-dimensional parameters show the ability to collapse the spread of data, in comparison to plotting $V_{\infty}$ and $\dot{m}_{\text{jet}}$. However, to better understand the distribution of the data in both sets of non-dimensional parameters, more data points that do not fall along the line are likely necessary. These points will likely fill in, and should be compared, when additional submerged jet cases are completed using a freestream liquid with varying density properties.

**Conclusions**

This chapter took both the data and regime classifications of Chapter 4 and investigated the creation of a submerged jet regime map. Each case was assigned a numeric value representing its regime, and this data resulted in a 2D spread of data over the input parameters ($V_{\infty}$ and $\dot{m}_{\text{jet}}$). Computer generated maps using linear and cubic interpretations left something to be desired, such as placing data points in the incorrect regimes, or creating ‘pocket’ regions of data. In addition, these interpolated maps caused very jagged transitions between regimes. Another map was created by visually splitting the data points, and produced smoother results. The conclusions drawn are that more data points are necessary, as well as utilizing different interpolation methods.

In addition to creating a regime map based on the input parameters of the simulations, two different non-dimensional values are used. Both sets of non-dimensional parameters resulted in the data collapsing, either to a cone or curve, both of which became a line when plotted on a log-log plot. These non-dimensional regime maps did resulting in separating the regimes to different locations along the log-log line of collapsed data, and are of interest into future work of submerged jet classification, particularly with different fluids for the freestream flow. The ability to predict what regime will form can be used to
use different regimes for different uses, such as avoiding impact with the nozzle by employing the Stable Cavity regime.

Recommended future work related to this chapter requires first collecting many more data points to fill out the observed regimes. Afterwards, more attempts should be taken at interpolated regime maps, both with different grid spacing and different interpolation methods. In addition, different methods to non-dimensionalize the data must be conducted. This will be especially useful when these regimes are applied to gas jets submerged in different fluids, such as an external mixture of air and gas. The changing density of the outer flow would be best visualized by being able to collapse all the controlling parameters into two or more non-dimensional parameters.
Chapter 6

Conclusions

The work in this thesis used CFD to evaluate several multiphase phenomenon. The focus of the study conducted in Chapter 2 was on the constant pressure assumption used in supercavity theory. This led to the characterization of distinct pressure regions within the cavity, which were evaluated using several methods to determine their effect on flow and cavity parameter prediction. The rest of the chapters switched focus to another multiphase setting, supersonic jets. Chapter 3 sets the scene of these jets, evaluating the effect of ambient liquid density on the structure of the gaseous jet. Chapters 4 and 5 then focus entirely on submerged gas jets with a significantly higher density liquid (air jetted into water). These chapters then varied the input parameters of freestream velocity and mass flow rate of the jet, which resulted in the classification of five distinct submerged jet regimes. These regimes were then evaluated in the context of creating a predictive regime map which can be used run submerged jet cases in a specific regime. A full breakdown of the information revealed in Chapters 2 through 5 is presented in the below sections.

Pressure Importance

Insight into pressure variations within ventilated cavities leads to a better understanding of the internal flow mechanisms of the cavities. These internal flows can lead to changes in cavity closure, and explain mechanisms behind air entrainment and gas flow patterns [3, 14]. The focus of this chapter was to understand if and how internal pressure variations occur, tie these pressure variations and gradients to flow character, and develop an understanding as to how they correspond to cavity shape. Employing CFD, multiple ventilated cavity simulations were run and utilized to provide detailed pressure data within the cavities. This pressure data, presented in terms of cavitation number ($\sigma$), revealed distinct regions of pressure that form within cavities. It was shown that these regions are different for different cavity closure
types, specifically twin-vortex and toroidal-vortex cavities. These regions seem to correspond to cavity shape, and are linked to regions of cavity radius change. Using the method of Gaussians, histograms of the pressure within the cavity were divided into a series of Gaussian distributions in order to better understand each peak. The distributions were combined using the Mixture-of-Gaussians approach to model the total distribution. These pressure variations were also investigated with respect to the internal cavity flow. By looking at the internal streamlines of the two cavity types, it is seen that both regimes contain recirculating internal flow within the cavity. Though the twin vortex cavity contains more flow that is attached to the interface and immediately ejected from the cavity, both the twin vortex cavities and toroidal cavities show signs of flow being redirected by the low cavitation number (high pressure) regions and controlled by internal pressure variations. This leads to the conclusion that better understanding these pressure regions within ventilated cavities is important for understanding internal flow dynamics. Lastly, the cavity pressure variations were investigated with respect to overall cavity shape and the semi-empirical relations between cavitation number and cavity dimensions. This is of the utmost importance as experimentally, pressure is usually taken as a single probe, however, the difference that can be seen in a single cavity can lead to upwards of 10% difference in cavity dimensions such as length and diameter. The assumption that cavity pressure is constant throughout the cavity was challenged at a detailed level. In general, the results indicate that the assumption is reasonable. For toroidal-vortex cavities, the assumption appears to be quite good. However, for twin-vortex cavities, the assumption remained reasonable but had some measurable consequences. Specifically, we observed that the impact leads to roughly a 10-15% uncertainty in the prediction of the cavity shape using semi-empirical theory. The overall implication is that such uncertainty should be considered when applying this constant-pressure assumption.
Submerged Jet Findings

The focus of Chapter 3 was to use CFD to improve the physical understanding of supersonic gas jets submerged into a liquid. The CFD was benchmarked against available data for a gas jet expelled into water. A mesh independence study indicated that the bulk properties of the jet were captured reasonably well. Although the mesh was found to slightly over-diffuse a gas-gas jet, in application to multiphase jets the dramatic radial expansion of the jet reduced concerns. Using this model setup, we developed an understanding of the impact of the external liquid density for both an over- and under-expanded jet.

Similar to previous experimental studies, the CFD model predicted that as the density of the ambient fluid increases, the gas-liquid interface becomes unstable and shorter jets occur. In addition, the “back-attack” [5,6] phenomenon (where the gas breaks down, develops a reverse flow, and approaches the nozzle exit) was observed in the CFD model. Chapter 3 work improves the understanding of the interactions of the gas-jet and the surrounding liquid, which is difficult to access visually from experiments.

Using the CFD model, details of the gas/liquid interactions were observed. We found that with increased liquid density, the turbulence level increased dramatically at the gas-liquid interface. This leads to a rapidly developing mixing layer that causes an earlier breakdown in the shock-cell structure. The overall result is a shorter penetration length of gas into the surrounding fluid. It was also found that this interaction is dominated by density, and becomes particularly important with a liquid density one-hundred times greater than the gas, observed from the extracted images.

Several novel characteristics were also observed in the CFD simulations. The first arises in the impact of the nozzle flow type. It was observed that the general characteristics of an over-expanded nozzle were nearly identical to the under-expanded nozzle. This indicates that the near-nozzle flow structure does not dominate the interaction. It was, however, observed that the gas-liquid interface could be destabilized earlier depending on this flow type, indicating that though the nozzle is not a dominating controller, it has a minor effect. Here it was observed that the under-expanded jet interface was
destabilized earlier than the over-expanded jet. The second novel observation involved the transmission of energy from the jet to the liquid. Due to the dramatic rise in turbulence, one would expect that the phases mix and diffuse energy. We also observed bulges of gas that act like a moving sphere. This moving sphere also imposed momentum transfer to the liquid.

Building off the ideas developed in Chapter 3, a more advanced round of parameters is investigated with respect to multiphase jets. Within Chapter 4, a 2D axisymmetric model for a submerged multiphase jet was validated and multiple cases were run. Liquid velocity and gas mass flow rate were varied to generate a comprehensive map of gas-liquid interactions for multiphase jets. This lead to the development of a regime map for these interactions, similar to those used in pipe flow. Five regimes were classified with respect to the current findings, which were Shedding Jet, Toroidal-Cavity, Pulsating Cavity, Stable Cavity, and Over-Ventilated Cavity. In addition to just characterizing these regimes based on visuals, the effect of these regimes and the varied parameters were also investigated with respect to the internal flow of the jets. The gas-liquid interactions cause expansions and contractions of the jet flow right out of the nozzle. The worry with this behavior is in chemical mixing purposes of jet use, where it can cause the reactions to take place on the jet and surrounding equipment, reducing the lifespan of such equipment. Similar behavior has been called attention to in the form of “back-attack” by Shi [6]. Away from the jet but within the cavities formed, the flow is very circulatory.

The understanding of multiphase jets, particularly submerged gas jets in a higher density fluid, are important to industrial uses. Chemical and thermal mixing, as well as nuclear reactors all depend on the dynamics of these interactions [4-6]. The more known about the internal and external properties of such systems, the better these systems can be both designed and maintained.

The study conducted in Chapter 5 is a continuation of the results discussed in Chapter 4. The regime classifications characterized in Chapter 4 were applied to all 64 run cases, producing an array of data for testing different regime creation techniques. The visually categorized regime cases were divided into a regime mapping using a computational linear interpolation, and computational cubic interpolation, and plain visual inspection. The conclusion between all methods is that the regimes seem to fan out from
the bottom left corner of the plot ($V_\infty$ and $m_{\text{jet}}$ approaching 0), causing the majority of the regime divisions to be diagonal. Another finding is that more cases are required to create a more detailed regime map using a computational method, as there is not appropriate data density to create consistently smooth lines which correctly place all the given data.

The regime data was also used with two different pairs of non-dimensional parameters. Both sets of parameters collapsed the data, with the data becoming a line on a log-log plot. However, this was not entirely useful, as it only clearly separated the Shedding Jet regime from the rest of the data. The information discovered in Chapter 5 forms the basis for future work both into overall regime mapping of submerged gas jets, as well as characterizing the regimes using non-dimensional parameters.

**Recommended Future Work**

Moving forward, there appears to be several ways to extend the present research into cavity pressure. Experimental verification of the present findings would be of high value. In addition, extending the MOG method to a range of conditions with a corresponding characterization of the variations would also be very useful. Lastly, extending these types of analyses to other cavity closure modes would also be of interest.

With regards to multiphase jets, there are many aspects that can be further studied to better understand this flow. Recommended future plans for this work are to continue to improve the numerical model by moving to higher resolution meshes that better resolve the far wake of the jet. In addition, developing an understanding of the initial interaction of the jet with the liquid is warranted to understand the cause of the earlier destabilization of the gas-liquid interface for the under-expanded jets in Chapter 3. In addition, the jet regimes of Chapter 4 can be further studied to better map the exact turning points between regimes. This would require a very large range of cases varying both liquid and gas conditions. To verify that the 2D axisymmetric modeling approach holds for all jets, it will also be important to conduct fully 3D simulations of the same data points.
Additional work that would help in the classification of jet regimes would be an advanced study using dynamical system methods, such as Proper-Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD). The frequency behavior of the different modes can also be analyzed using Fast Fourier Transform (FFT). The ability to describe the regimes using only two non-dimensional parameters is another area worth investigating further, especially with respect to including other freestream liquids.
Appendix A: Effect of Ambient-Fluid Viscosity and Compressibility

In this section, we developed sensitivity studies to rank the primary factors of the influence of a liquid on a gas jet. It was found that the primary parameter is the ambient fluid density, $\rho_L$. We also evaluated the impacts of liquid compressibility and liquid viscosity, which were much less sensitive. The study used a light water case, which assumes an incompressible liquid with a $\rho_L$ value of 1 kg/m$^3$ and a dynamic viscosity, $\mu_L$, of 8.887e-4 Pa-s. Results from this case are compared to a case with ‘air’ as the surrounding liquid, with compressibility (based on air properties) and a $\mu_L$ value of 1.855e-5 Pa-s. Results from these cases are presented in Figure A1 and Figure A2 below. In Figure A1, is a plot of $M$ as a function of distance downstream of the nozzle exit. The results show a similar character despite large differences in both the ambient-fluid compressibility and viscosity. The contour plots in Figure A2, comparing several flow variables, qualitatively verify that neither compressibility nor viscosity drive the jet-character when submerged in liquid. From this result, it is concluded that the differences between the air and the light water case is minimal and that density is the main driver of the interaction.

![Figure A1: Local M plotted against axial location](image)
Figure A2: Comparison of an air-jet released into compressible air to an air-jet released into a light water, shown with the same color scales.
Appendix B: Additional Under Expanded Jet Cases

In this appendix is the complete presentation of the CFD results for the under-expanded, NPR=0.08, nozzle condition. Figure B1 is a contour of the local Mach number. Similar to the over-expanded case; the M contours indicate that the jet diffuses more with an increased value of $\rho_L$. This conclusion is consistent with the over-expanded condition.

![Figure B1: M at varying at different $\rho_L$ values and 0.08 NPR](image)

TVR is plotted for the under-expanded jet condition in Figure B2. The conclusions for the TVR behavior are completely consistent with the over-expanded jet. That is, the turbulence levels rise with $\rho_L$ values and are locally high at the gas-liquid interface.

![Figure B2: TVR of the jet at different $\rho_L$ values and 0.08NPR](image)
A contour plot of $p$ is shown on the jet centerline in Figure B3. As expected, the general structure of the pressure fluctuations is observed in the supersonic regions of this under-expanded jet. Note that the right-most pressure fluctuations are outside of the plotted domain. In general, the flow remains consistent with the over-expanded condition.

Figure B3: Pressure at different $\rho_L$ values and 0.08 NPR
# Bibliography

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
</tbody>
</table>