MODELING LASER-GENERATED CAVITATION BUBBLES

A Thesis in
Mechanical Engineering
by
Cinnamon M Christian

© 2012 Cinnamon M Christian

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Master of Science

August 2012
The thesis of Cinnamon M Christian was reviewed and approved* by the following:

Eric G. Paterson  
Professor of Mechanical Engineering  
Senior Scientist, Applied Research Lab  
Thesis Co-Advisor

Arnold A. Fontaine  
Senior Scientist, Asst. Dir., Applied Research Lab  
Thesis Co-Advisor

Anil K. Kulkarni  
Professor of Mechanical Engineering

Karen A. Thole  
Professor of Mechanical Engineering  
Head of the Department of Mechanical Engineering

*Signatures are on file in the Graduate School.
Abstract

Cavitation erosion is a flow phenomenon where the collapse of cavitating vapor bubbles against a solid surface leads to the erosion of the surface. Cavitation affects various types of turbomachinery, including impeller blades, valves and ship propeller blades, and can affect both the performance and the life-cycle of a machine. A significant amount of research regarding cavitation erosion has focused on the formation and collapse of a single bubble. In experimental research a laser is often used to generate a bubble near a solid surface; mimicking the flow dynamics of cavitation erosion. An experimental setup was devised where a single cavitation bubble was generated using a Nd:YAG laser. A thermal analysis of the formation of a laser-generated cavitation bubble was conducted using the open source CFD package OpenFOAM to model the heating of the water by the laser. The collapse of a laser-generated cavitation bubble was also modeled computationally using a compressible multiphase finite volume method.

It was determined experimentally that a typical laser-generated cavitation bubble has a maximum radius on the order of 1 mm. A single bubble with a maximum radius of 1 mm and a collapse time of approximately 190 microseconds was used to model the size and shape of a laser-generated cavitation bubble. Using CFD the collapse of a single bubble in a free field was analyzed and compared to the solution of the Rayleigh-Plesset equation, as well as experimental data. The collapse of a single bubble against a solid wall was then modeled, varying the standoff distance to the wall in order to observe the different bubble dynamics discussed in the literature. The compressible multiphase flow method successfully modeled the dynamics of the bubble collapse; including both the high pressure pulse emitted upon collapse as well as the microjet that forms during collapse and travels from the center of the bubble towards the wall.
## Table of Contents

List of Figures vii  
List of Tables ix  
List of Symbols x  
Acknowledgments xiv  

### Chapter 1  
**Introduction**  
1.1 Motivation .................................................. 1  
1.2 Literature Review ............................................. 2  
1.2.1 Cavitation Erosion ......................................... 2  
1.2.1.1 Cavitation Erosion in Fluid Machinery .................... 2  
1.2.1.2 Experimental Research on Cavitation Damage ................. 3  
1.2.1.3 CFD Cavitation Erosion Models ........................... 5  
1.2.2 Single Bubble Collapse .................................... 6  
1.2.2.1 Methods for Generating a Single Bubble .................. 6  
1.2.2.2 Laser-Generated Cavitation Bubbles ....................... 7  
1.2.2.3 Single Bubble Collapse Near a Wall ...................... 8  
1.2.3 CFD of Single Bubble Dynamics ............................. 10  
1.2.3.1 Modeling Single Bubble Collapse ......................... 10  
1.2.3.2 Single Bubble Collapse Near a Solid Boundary ............. 13  
1.3 Objective .................................................. 15  
1.4 Agenda ................................................... 15  

### Chapter 2  
**Methodology**  
2.1 Approach .................................................. 16  
2.2 CFD of Laser Heating ......................................... 16
List of Figures

1.1 Cavitation damage of a centrifugal pump impeller . . . . . . . . . . . . 3
1.2 A laser-generated cavitation bubble with $R_{\text{max}} = 1.45$ mm . . . . . 8
2.1 Schematic of laser-generated cavitation bubble experiment . . . . . . . 17
3.1 $R/R_{\text{max}}$ vs $t/\tau_c$ calculated using the Rayleigh-Plesset equation . . 33
3.2 Radius vs time for different pressure ratios . . . . . . . . . . . . . . . . 35
3.3 Nondimensional radius vs nondimensional time for various pressure ratios 35
3.4 The compression ratio vs the pressure ratio . . . . . . . . . . . . . . . . 37
3.5 Nondimensional time vs nondimensional radius for different $R_{\text{max}}/R_0$ . 38
4.1 Beam diameter of the laser . . . . . . . . . . . . . . . . . . . . . . . . 41
4.2 Axisymmetric wedge mesh for CFD of laser heating . . . . . . . . . . 42
4.3 Block numbering of the wedge mesh for CFD of laser heating . . . . . 42
4.4 The boundaries of the wedge mesh for CFD of laser heating . . . . . . 43
4.5 The absorption coefficient vs the percentage of particulates . . . . . . 48
4.6 Temperature vs the absorption coefficient at different laser energy outputs 50
4.7 Locations in the mesh where temperature is being sampled . . . . . . . 53
4.8 Temperature distribution of the laser focal volume at different times . . 53
4.9 Point distribution of the temperature in the focal volume at different times 54
4.10 Contour plot of the temperature in the laser focal volume at $t = 10$ ns . 54
5.1 Axisymmetric wedge mesh for CFD of single bubble collapse . . . . . . 58
5.2 Mesh distribution of wedge mesh for single bubble collapse . . . . . . 58
5.3 Mesh generation for bubble collapse against a wall . . . . . . . . . . . 60
5.4 Boundary patches for CFD of single bubble collapse . . . . . . . . . . 62
5.5 Boundary conditions for bubble collapse under a plate . . . . . . . . . 63
5.6 Boundary conditions for bubble collapse above a plate . . . . . . . . . 63
5.7 Radius vs time for the formation and collapse of a single bubble . . . . 66
5.8 Nondimensional radius vs nondimensional time for a single bubble . . . 67
5.9 Volume fraction of the gas phase of a single bubble at different times . . 68
5.10 Pressure during the formation and collapse of a single bubble . . . . . 69
5.11 Comparison between CFD and the Rayleigh-Plesset equation . . . . . 70
5.12 Comparison with data for $P_{G_0} = 52.1$ MPa and $R_0 = 62.9 \, \mu m$ 71
5.13 Comparison with data for $P_{G_0} = 2330$ Pa and $R_0 = R_{max} = 746.9 \, \mu m$ 72
5.14 Comparison with data for $P_{G_0} = 4.579$ Pa and $R_0 = R_{max} = 746.9 \, \mu m$ 73
5.15 Bubble radius vs time with t1, t2 and t3 indicated 74
5.16 Volume fraction of the gas vs $r$ for three different times 76
5.17 Pressure vs $r$ for three different times 77
5.18 Pressure at different locations for different standoff distances 79
5.19 Volume fraction of the gas during bubble collapse against a wall for $\gamma = 0.2$ 80
5.20 Pressure during bubble collapse against a wall for $\gamma = 0.2$ 81
5.21 Change in pressure during bubble collapse against a wall for $\gamma = 0.2$ 82
5.22 Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 0.2$ 83
5.23 Volume fraction of the gas during bubble collapse against a wall for $\gamma = 0.5$ 84
5.24 Pressure during bubble collapse against a wall for $\gamma = 0.5$ 85
5.25 Change in pressure during bubble collapse against a wall for $\gamma = 0.5$ 86
5.26 Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 0.5$ 86
5.27 Volume fraction of the gas during bubble collapse against a wall for $\gamma = 0.7$ 87
5.28 Pressure during bubble collapse against a wall for $\gamma = 0.7$ 88
5.29 Change in pressure during bubble collapse against a wall for $\gamma = 0.7$ 89
5.30 Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 0.7$ 89
5.31 Volume fraction of the gas during bubble collapse against a wall for $\gamma = 1.0$ 90
5.32 Pressure during bubble collapse against a wall for $\gamma = 1.0$ 91
5.33 Change in pressure during bubble collapse against a wall for $\gamma = 1.0$ 92
5.34 Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 1.0$ 92
5.35 Volume fraction of the gas during bubble collapse against a wall for $\gamma = 1.3$ 93
5.36 Pressure during bubble collapse against a wall for $\gamma = 1.3$ 94
5.37 Change in pressure during bubble collapse against a wall for $\gamma = 1.3$ 94
5.38 Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 1.3$ 95
5.39 Maximum pressure for different $\gamma$ 96
5.40 Maximum jet velocity for different $\gamma$ 97
5.41 Wall shear stress along the wall for different $\gamma$ 98
5.42 Velocity of the microjet for bubble collapse at $\gamma = 0.7$ and $\gamma = 1.2$ 99
5.43 The effect of gravity on the volume fraction of the gas during collapse 101
5.44 The effect of gravity on pressure during bubble collapse 102
5.45 Pressure versus $x$ at $y = s$ and 80 $\mu s$, 100 $\mu s$ and 120 $\mu s$ 103
## List of Tables

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Table Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Time scales of the Rayleigh-Plesset equation for different $R_0$</td>
<td>32</td>
</tr>
<tr>
<td>3.2</td>
<td>Fluid parameters for the solution of the Rayleigh-Plesset equation</td>
<td>32</td>
</tr>
<tr>
<td>3.3</td>
<td>Parameters for the Rayleigh-Plesset equation at different pressure ratios</td>
<td>34</td>
</tr>
<tr>
<td>3.4</td>
<td>Results of the Rayleigh-Plesset equation using different $R_0$</td>
<td>37</td>
</tr>
<tr>
<td>4.1</td>
<td>Number of cells in mesh for CFD of laser heating</td>
<td>43</td>
</tr>
<tr>
<td>4.2</td>
<td>Boundary conditions for CFD of laser heating</td>
<td>44</td>
</tr>
<tr>
<td>4.3</td>
<td>Fluid parameters for CFD of laser heating</td>
<td>44</td>
</tr>
<tr>
<td>4.4</td>
<td>Laser parameters for CFD of laser heating</td>
<td>45</td>
</tr>
<tr>
<td>4.5</td>
<td>Experimental data from absorption coefficient test</td>
<td>47</td>
</tr>
<tr>
<td>4.6</td>
<td>Numerical schemes for CFD of laser heating</td>
<td>49</td>
</tr>
<tr>
<td>4.7</td>
<td>Grid refinement study: comparison of the temperature</td>
<td>51</td>
</tr>
<tr>
<td>4.8</td>
<td>Time study: comparison of the temperature</td>
<td>52</td>
</tr>
<tr>
<td>5.1</td>
<td>Mesh distribution for different standoff distances $\gamma$</td>
<td>61</td>
</tr>
<tr>
<td>5.2</td>
<td>Boundary conditions for CFD of single bubble formation and collapse</td>
<td>61</td>
</tr>
<tr>
<td>5.3</td>
<td>Boundary conditions for CFD of bubble collapse against a wall</td>
<td>62</td>
</tr>
<tr>
<td>5.4</td>
<td>Gas parameters for CFD of single bubble formation and collapse</td>
<td>64</td>
</tr>
<tr>
<td>5.5</td>
<td>Liquid parameters for CFD of single bubble formation and collapse</td>
<td>64</td>
</tr>
<tr>
<td>5.6</td>
<td>Numerical schemes for CFD of single bubble formation and collapse</td>
<td>65</td>
</tr>
<tr>
<td>5.7</td>
<td>Initial conditions for CFD of single bubble collapse</td>
<td>66</td>
</tr>
<tr>
<td>5.8</td>
<td>$R_{\text{max}}$ and $\tau_C$ for CFD and the Rayleigh-Plesset equation</td>
<td>70</td>
</tr>
<tr>
<td>5.9</td>
<td>CFD and experimental jet velocity for $\gamma = 0.7$ and $\gamma = 1.2$</td>
<td>99</td>
</tr>
<tr>
<td>5.10</td>
<td>CFD and experimental impact pressure for different standoff distances</td>
<td>100</td>
</tr>
</tbody>
</table>
List of Symbols

\begin{itemize}
    \item \(a\) Laser rise time
    \item \(a_c\) Adiabatic constant
    \item \(a_w\) Absorption coefficient
    \item \(A\) Source term for laser heating
    \item \(b\) Confocal parameter
    \item \(c\) Speed of sound
    \item \(c_p\) Heat capacity at constant pressure
    \item \(c_v\) Heat capacity at constant volume
    \item \(C_0\) Courant number
    \item \(D\) Strain rate tensor
    \item \(D_T\) Thermal diffusivity
    \item \(E_p\) Energy of laser pulse
    \item \(f\) Focal length of a lens
    \item \(f(r, x)\) Laser spatial profile
    \item \(g\) Gravitational acceleration
    \item \(i, j, k\) Indices
    \item \(I\) Laser intensity (or irradiance)
    \item \(I_0\) Initial laser intensity
    \item \(I_m\) Maximum laser intensity
\end{itemize}
$k$  Thermal conductivity
$L$  Focal length of laser beam
$m_p$  Total mass of seeding particles
$p$  Pressure
$p_0$  Reference pressure
$p_B$  Pressure inside bubble
$p_d$  Piezometric pressure
$p_{G_0}$  Initial gas pressure inside bubble
$p_v$  Vapor pressure
$p_{\infty}$  Pressure far away from the bubble
$q(t)$  Laser temporal profile
$r$  Radial coordinate
$R$  Bubble radius
$R_0$  Initial bubble radius
$R_{gas}$  Individual gas constant
$R_w$  Reflectivity
$Re$  Reynolds number
$S$  Surface tension
$t$  Time
$T$  Temperature
$T_\infty$  Temperature far away from the bubble
$u, v, w$  Velocity components in Cartesian coordinates
$u_r, u_\theta, u_\phi$  Velocity components in spherical coordinates
$v_{imp}$  Impulse velocity of the microjet
$v_{max}$  Maximum velocity of the microjet
$V$  Volume
$V_{focal}$  Focal volume of the laser beam
$V_{pt}$  Total volume of seeding particles
$V_w$  Volume of water
$w$  Laser radius
$w_0$  Waist size
$w_f$  Beam radius at the focal lens
$z_0$  Rayleigh length
$\alpha_1$  Volume fraction of the gas
$\alpha_2$  Volume fraction of the liquid
$\gamma$  Standoff distance
$\kappa$  Ratio of specific heats
$\lambda$  Wavelength of laser
$\mu$  Dynamic viscosity
$\mu_{eff}$  Effective viscosity
$\mu_L$  Dynamic viscosity of a liquid
$\nu$  Kinematic viscosity
$\nu_L$  Kinematic viscosity of a liquid
$\rho$  Density
$\rho_0$  Reference density
$\rho_1$  Density of gas in a two-phase flow
$\rho_2$  Density of liquid in a two-phase flow
$\rho_L$  Density of a liquid
$\rho_p$  Density of a seeding particle
$\sigma_{ij}$  Stress tensor
$\tau_c$  Collapse time
$\tau_p$  Pressure time
\(\tau_s\) Surface tension time
\(\tau_w\) Wall shear stress
\(\tau_\nu\) Viscous time
\(\phi\) Volume fraction of seeding particles in water
Acknowledgments

I would like to begin by thanking my research advisors Dr. Arnold Fontaine and Dr. Eric Paterson, who have provided me with an excellent educational experience. I am very grateful for their support and guidance. I also want to thank Sandia National Laboratories for funding this research project.

In addition, I want to thank my friends, classmates and labmates who have made graduate school such a rewarding experience.

I want to thank my parents and siblings for their constant support and encouragement, and finally I want to thank Dru, for always listening.
Introduction

1.1 Motivation

Cavitation bubbles form in liquid when the pressure of the liquid drops below the saturated vapor pressure, this causes a bubble to form, expand and quickly collapse resulting in the emission of a high pressure pulse [1]. Cavitation affects a wide range of industries and can affect the performance of a machine, lead to significant noise and vibration issues and cause damage to solid boundaries [2]. Cavitation can be a useful flow phenomenon, it is often used in applications such as ultrasonic cleaning and industrial cleaning [2]. However, it can also be a potentially damaging flow phenomenon, such as when it occurs in fluid machinery. When vapor cavities become entrained by the flow they can collapse near solid boundaries, causing damage [3]. The damage caused by cavitation is referred to as cavitation damage, or cavitation erosion; it occurs when a cavitation bubble collapses near a solid boundary, causing an erosion of the material. It effects various turbomachinery and hydraulic equipment, including impeller blades, valves, and ship propeller blades [4]. An understanding of cavitation damage is important because cavitation has such an influence on a wide range of machinery and industries, and can have irreversible consequences on the performance and life-cycle of the machinery that it effects.

Research on cavitation erosion has focused both on empirical methods that can be used in industry to predict and avoid cavitation damage, and single bubble dynamics
that can be used to explain the phenomenon of cavitation bubble collapse. While a cavitating flow contains numerous bubbles, many researchers have for simplicity studied only the formation and collapse of a single bubble. A single bubble can be produced by a laser, which can precisely focus a beam of light into a liquid causing a bubble to form and collapse \[5\]. This method has been used by numerous researchers to analyze the dynamics of bubble collapse, which is why it is of interest to study both the formation and collapse of a laser-generated cavitation bubble.

### 1.2 Literature Review

#### 1.2.1 Cavitation Erosion

Hydrodynamic cavitation erosion is a high speed flow phenomenon that commonly occurs in fluid machinery \[3\]. Cavitation is common in high speed flows where the pressure of the liquid drops below the vapor pressure; causing a vapor cavity to form. The collapse of these cavities against solid surfaces leads to damage of the surface. The primary effect of cavitation on fluid machinery is loss of efficiency; however in extreme cases cavitation damage can eventually lead to complete failure of the machinery \[2\]. As cavitation damage primarily affects turbomachinery, a significant amount of research has focused on what types of metals and alloys best resist cavitation damage. To perform this type of analysis four tests are primarily used: the Venturi method, the rotating disk method, the cavitating jet method and the vibratory test \[6\]. Some numerical simulations \[7, 8\] have also been performed that attempt to measure how much material will be eroded by exposure to cavitating flows using different methods to account for the fluid-structure interaction.

#### 1.2.1.1 Cavitation Erosion in Fluid Machinery

Cavitation is an important design criteria in fluid machinery such as pumps, turbines, and propulsion systems; all of which are often subjected to local high speed flows and are subsequently susceptible to cavitation. These types of fluid machinery are typically designed to either have no cavitation or a minimal amount of cavitation. In pumps and
turbines cavitation causes a decrease in efficiency, and in propulsion systems cavitation can decreases the thrust produced by the system [2].

Figure 1.1: Cavitation damage of a centrifugal pump impeller; example of a cavitating flow and the damage caused by cavitation [1]

Depending on the design of the machine, cavitation damage can range from a small amount of pitting to the failure of the machine. Complete failure of the machinery is often preceded by issues such as leakage, unwanted mixing of fluids, and an overall decrease in the performance of the system [9]. Cavitation can also impact newer technologies such as hydrokinetic turbines, where cavitation can occur on rotating turbine blades that are submerged in water [10]. Despite the consequences of cavitation damage in fluid machinery, it is often unavoidable because of the importance of other design criteria such as operating conditions and economic constraints [2]. A better understanding of the cavitation erosion process is therefore essential for optimizing cavitation design in fluid machinery.

1.2.1.2 Experimental Research on Cavitation Damage

There are a variety of experimental methods commonly used for testing cavitation damage in materials. The most common of which include the Venturi method, the rotating disk method, the cavitating jet method and the vibratory test. The Venturi method uses either a Venturi nozzle, or a section of a water tunnel, to impose a cavitating flow upon
a test section [6]. Of all of the methods it has been argued that the Venturi method most closely models the actual flow conditions under which cavitation erosion occurs [6, 11]. Despite the advantage of being able to closely emulate actual field conditions, this method requires a large-scale testing facility that is costly to build and operate [6, 11]. The rotating disk method uses a spinning disk perforated with small holes to create cavitation bubbles in a fluid [6]. The primary advantage to using the rotating disk method is that it most accurately simulates the flow conditions that occur in pumps [12]. Conversely the biggest disadvantage is that while it generates a complex flow that closely models the conditions of certain rotating devices, the flow pattern is not indicative of all cavitating flows [6]. The cavitating jet method consists of imposing a high pressure jet into a low pressure chamber; vortex cavitation forms on the outer edges of the jet, simulating cavitation damage as the cloud cavities form and collapse against a surface placed downstream of the jet [6]. This method has been successfully used to study cavitation erosion in valves and hydraulic equipment where the impingement of a cavitating jet is the primary cause of damage; it has been found that the distance between the jet impingement and the specimen is important to the type of damage observed [13]. While all three of the methods discussed so far have their merits, the most common method for studying cavitation damage in materials is the vibratory test [6, 14].

The vibratory test, often referred to as an oscillating horn, is a test where a small specimen is submerged in a liquid and subjected to an intense and rapid oscillation of the pressure using a magnetostriction device, causing the liquid to cavitate [6, 14]. The primary advantage of the vibratory test is that it is relatively simple and inexpensive to run, and of shorter duration relative to the other types of erosion testing [6, 14]. This method has been used primarily to test the resilience of certain materials to cavitation damage. Feller et al. used a vibratory apparatus to test various commercial metals, as well as several high purity metals [15]. Using the same testing conditions for each metal they weighed the sample before and after the test was completed, as well as analyzed the sample using a scanning electron microscope [15]. Okada et al. used a vibratory test to directly study cavitation erosion by measuring the pressures generated upon the specimen by bubble collapse [16]. They measured the impact loads generated by collapsing bubbles
using a piezoelectric ceramic disk, and found that the magnitude of the pressure was
dependent on the size, shape and location of the bubble [16]. The primary disadvantage
of the vibratory method is that there is no direct correlation between the length of a test
run and the equivalent length of time necessary for that specimen to achieve the same
amount of damage in a field setting [6, 14]. Despite this limitation, the vibratory test
remains a popular method for testing cavitation damage because it is cost-efficient and
easy to standardize.

1.2.1.3 CFD Cavitation Erosion Models

As a result of the high cost associated with most experimental methods designed to
measure cavitation erosion an emphasis has been made on numerical simulations that
model the behavior of cavitating flows. Besides the significant body of work regarding
cavitating and boiling flows, a few researchers have focused on cavitating flows with an
emphasis on modeling cavitation erosion. There have been several different ways this
has been approached. Fortes-Patella et al. tried to predict the level of cavitation damage
using the Keller model, which is similar to the Rayleigh-Plesset equation except that it
accounts for a slight compressibility in the fluid. The pressure waves emitted upon the
collapse of the bubble were determined from the Keller model and then coupled with a
finite element code that solved the conservation of mass equation, the fundamental law
of dynamics, and an additional constitutive equation [7]. Although the Keller model,
like the Rayleigh-Plesset equation, is a simplified model, Fortes-Patella et al. were able
to model the fluid-solid interactions between bubble collapse and a solid boundary.

A similar approach was taken by Dular et al. who combined a CFD analysis with an
erosion model in order to capture both the flow dynamics as well as predict cavitation
erosion [8]. The CFD component of the model consisted of a homogeneous flow model
where the multiphase flow was modeled using the conservation equations for single-phase
flow with the addition of a barotropic state law to account for the difference in densities
between the two phases [8]. The erosion model assumes that cavitation cloud collapse on
a solid boundary leads to the formation of a shock wave and the subsequent formation
of a high speed micro-jet; the speed, size and impact pressure of the micro-jet were
calculated using analytical relationships and compared to the yield stress of the material [8]. While Dular et al. successfully coupled a CFD model with a cavitation erosion model, their approach relied on the assumption that the driving factor of cavitation damage is the formation of a micro-jet, an assumption that is still the subject of much debate and analysis.

1.2.2 Single Bubble Collapse

Single bubble collapse has become a research area of interest in an attempt to fully understand the complex cavitation erosion process and its possible mechanisms. After creating a single cavitation bubble through various means; such as spark generation, laser generation or acoustic generation, most researchers observed the bubble collapse through some combination of high speed photography or Schlieren visualization methods [17]. Since the goal of such research is to learn more about the physics of bubble collapse and how it affects cavitation erosion, a single bubble is often generated near a solid boundary in order to observe the effect the collapse of the bubble has on a solid surface [4, 17, 18, 19, 20, 21, 22]. High-speed photography and advanced optical techniques have shown that when a bubble collapses near a surface a large localized shock wave is emitted, and a high-speed jet travels towards the surface [17]. The debate over which of these phenomena is primarily responsible for cavitation damage is a long one; however, both phenomena are indicative of bubble collapse [1].

1.2.2.1 Methods for Generating a Single Bubble

There are several methods for experimentally producing a single bubble; using a laser, acoustic methods or spark generation are three of the most common methods for generating individual bubbles. Spark generation is a method where a spark is formed between two submerged electrodes, creating a void where a cavitation bubble can form [5, 19]. One of the issues with this method is that the electrodes can disturb the flow, causing perturbations that can lead to inaccuracies in the data [5, 19]. This is the primary reason that using lasers to generate single bubbles has become such a common method for studying cavitation bubbles [19]. Acoustic methods for generating cavitation bub-
bles are also common and include a method where a bubble is produced in an acoustic trap [5]. Acoustic methods are widely used when studying acoustic cavitation, however, when studying bubble collapse and cavitation damage, bubbles are typically generated by lasers.

1.2.2.2 Laser-Generated Cavitation Bubbles

Using laser-generated cavitation bubbles as a means of studying cavitation erosion has been a common method since Lauterborn et al. investigated bubble collapse near a solid boundary in 1975 [20]. Since then many researchers have contributed to the understanding of bubble dynamics and cavitation using this technique. Typically there is a lens system which focuses the energy from the laser into a small focal volume; causing local heating of the water. The amount of heating is dependent on the energy of the laser beam, which depends primarily on the type of laser used, the focusing angle, and the laser pulse duration [23]. A Nd:YAG laser is often used [17, 18, 19], which depending on the laser can have pulse durations between nanoseconds and femtoseconds [23]. The phenomenon itself occurs on a very small scale as well; for a cavitation bubble with a maximum radius of 1 mm, the collapse time of the bubble can be estimated using the Rayleigh equation to be on the order of 100 microseconds [1]. However, despite the small size of the bubble, and the short duration of the collapse, the pressures generated by the bubble collapse are very large, approximately on the order of 100 MPa [17].
1.2.2.3 Single Bubble Collapse Near a Wall

The collapse of a single bubble near a solid boundary has also been extensively studied, as this experimental setup closely mimics the cavitation erosion phenomenon. While most experimentalists have focused on using lasers to create bubbles near a solid boundary, Tomita et al. used spark generation to look at bubble collapse near three different solid boundaries [21]. The three boundaries that were used included a solid boundary with a pressure transducer to measure the pressure distribution, a solid boundary comprised of photoelastic material which was used to observe the local stress rate and a solid boundary comprised of soft material to observe the damage caused by bubble collapse [21]. Tomita et al. argued that four distinct pressures are generated during the bubble collapse; the pressure and shock waves generated during the collapse of the bubble and the bubble rebound, the impact pressure generated by the jet that forms as the bubble collapses against a surface, and the pressure generated by the collapse of many tiny bubbles that form after the jet impacts the surface [21]. Tomita et al. also observed that the distance of the bubble to the boundary was a critical parameter that affected the damage patterns left by the bubble collapse [21].

After realizing the critical importance of the proximity of the collapsing bubble to the
solid surface, many researchers observed that the distance between the bubble and the wall affected not only the damage pattern but the magnitude of the pressure and velocity fields at collapse [17, 18, 19, 21]. The parameter usually used to define this distance is called the standoff distance. The standoff distance is defined as $\gamma = s/R_{max}$, where $R_{max}$ is the maximum radius of the bubble and $s$ is the distance between the center of the bubble at formation and the solid surface [17]. Philipp and Lauterborn used a Nd:YAG laser and a high-speed camera to produce and record the collapse of a bubble near a solid boundary [17]. They looked extensively at standoff distance and its effect on the collapse of the bubble. They determined that damage only occurred at standoff distances of two or less, and found that a jet only causes damage when the bubble is very close to the boundary; at a standoff distance equal to or less than 0.7 [17]. They observed that the boundary was only affected by the first collapse of the bubble at standoff distances less than one, but that the boundary was affected by the second bubble collapse at standoff distances less than two [17]. They argued that the greatest damage occurred when the bubble was very close to the boundary, at standoff distances less than or equal to 0.3, as well as when the bubble was farther away, at standoff distances between 1.2 and 1.4 [17].

A theory for this conclusion was put forth by Brujan et al. who also used a Nd:YAG laser to look at the collapse of a laser-generated cavitation bubble near a rigid boundary [18]. In addition to collecting experimental data on bubble collapse, Brujan et al. also used the Boundary Integral Method to numerically compute the pressure and velocity fields during bubble collapse [18]. They argued that the damage might be reduced for certain standoff distances, for example between 0.5 and 1.1, because of the contact between the bubble wall and the solid surface during the first collapse; which attenuates the bubble collapse and decreases the magnitude of the shock wave emitted [18]. Besides affecting the magnitude of the shock wave, the standoff distance has also been shown to affect the jet velocity as well as the dynamics of the high-speed jet formed during collapse [22]. The standoff distance has an effect on almost every aspect of the bubble collapse and the resulting damage, it is therefore an important parameter in research on bubble collapse.
1.2.3 CFD of Single Bubble Dynamics

Since a significant portion of research regarding cavitation erosion focuses on single bubble dynamics, it was of interest to find previous research that focused on computationally modeling single bubble dynamics. Single bubble dynamics are important not only in research regarding cavitating flows, but also in boiling flows and other various two-phase flows. Simplified methods of computationally studying bubble dynamics include the Rayleigh-Plesset equation, and its many variations, as well as the Boundary Integral Method. More recent computational research has focused mainly on using CFD to model bubble dynamics.

1.2.3.1 Modeling Single Bubble Collapse

Single bubble dynamics are typically modeled using a two-phase flow model; many CFD codes use the conservation of mass, momentum and energy equations as well as an additional equation of state for each phase. In most of the early CFD research on single bubble collapse, two-phase flow models were used but it was assumed that the flow was incompressible. Sussman et al. used a coupled level set and volume-of-fluid method to solve the incompressible mass and momentum conservation equations to get the pressure and velocity fields for the growth and collapse of a vapor bubble [24]. Sussman et al. assumed that the gas inside of the bubble behaved as a perfect adiabatic gas [24]. The main drawback with assuming that the liquid is incompressible is that compressibility effects become important during bubble collapse.

More recent research has used compressible two-phase CFD codes to model single bubble collapse. For example, Johnsen and Colonius used the unsteady Euler equations and an equation of state to model the collapse of a non-spherical bubble [25]. A similar approach was taken by Müller et al. who also assumed that the collapse was both compressible and inviscid; using the Euler equations and the stiffened gas law to model bubble collapse [26]. Both of these models were homogeneous models; a model which assumes that the two phases are in thermodynamic equilibrium with each other at the phase interface [27]. For a laser-generated cavitation bubble however, this is not a good assumption, as the rapid temperature rise of the liquid is what causes the bubble to
Only a few researchers have focused directly on modeling the collapse of laser-generated cavitation bubbles. Akhatov et al. developed a mathematical model for laser-generated bubbles that included the effects of the compressibility of the liquid, the heat and mass transfer between the phases, and the evaporation and condensation of the fluid at the bubble wall [19]. They developed a system of partial differential equations for a compressible multiphase flow using the mass, momentum and energy equations, with evaporation and condensation accounted for in the boundary conditions, and equations of state for the liquid and the gas [19]. Müller et al. used experimental data taken from laser-generated bubbles to estimate the maximum size of the bubble and the initial state of the gas inside the bubble in order to initialize the simulation of the bubble collapse [26]. They then used a compressible multiphase code and compared the results using both the Saurel-Abgrall approach, with the phase indicator as the gas fraction, and the real ghost fluid method, where the phase indicator was the level set function, and then compared them to experimental data [26].

Instead of solving the compressible mass, momentum and energy equations using CFD, Dreyer et al. took an analytical approach and created a set of ordinary differential equations that could be solved using Runge-Kutta methods under different assumptions to model laser-generated cavitation bubbles [28]. Dreyer et al. looked at the effects of compressibility, the effect of adding a phase change model that accounted for evaporation and condensation, and the effects of surface tension and heat transfer [28]. The result of their analysis showed that if the pressure surrounding the liquid remains at atmospheric pressure, as with a laser-generated cavitation bubble, then the temperature is going to change across the liquid-vapor interface and a phase change model becomes essential to accurately modeling the temperature distribution of a laser-generated bubble [28].

A non-equilibrium model is typically used for two-phase flows where the assumption cannot be made that the flow is in thermal equilibrium, this model is a type of phase change model. There are typically two approaches to modeling a two-phase system such as boiling or cavitation; employ a complete two-phase flow model with a set of transport equations for each phase or use a pseudo-fluid approach where one set of governing
equations is used and a density variable accounts for the mixture of the two phases [29]. The first approach was used by Zein et al. to developed a compressible two-phase non-equilibrium model for transition flow [30]. This model included seven equations; solving for velocity, pressure and temperature in both phases, and included heat and mass transfer in order to account for the phase change [30].

Zein looked specifically at using this seven equation multiphase model to numerically simulate the collapse of a laser-generated cavitation bubble [31]. Zein found fairly good agreement with the experimental results of Müller et al. [26] using a phase change model and assuming that the bubble contained non-condensable gas [31]. Zein [31] verified the conclusions of Dreyer et al. [28], who found that when the bubble was modeled as being completed filled with vapor at its maximum, a rebound bubble would not form if mass transfer was included. Experimentally, a rebound bubble is always seen with the collapse of a laser-generated cavitation bubble, therefore both Zein [31] and Dreyer et al. [28] chose to include a small percentage of non-condensable gas in the bubble. Dreyer et al. assumed that the non-condensable gas was comprised of hydrogen and oxygen [28], while Zein experimented with changing the mass fraction of the gases and found that it does affect the size and shape of the rebound bubble [31]. While the approach of Zein [31] is perhaps the most complete method for numerically modeling the collapse of a laser-generated cavitation bubble, it is a complicated set of equations to implement and is often very computationally expensive. Additionally, Dreyer et al. found that a phase change model has no significant affect on the evolution of the bubble radius, although it does significantly affect the temperature [28].

An alternative approach in non-equilibrium models is to use one set of transport equations for both phases and an additional relationship that accounts for non-equilibrium heat transfer. Schmidt et al. used such an approach, using the Homogeneous Relaxation Model as an additional relationship modeling the heat and mass transfer between the two phases in non-equilibrium channel flow [29]. Kunkelmann et al. also used an incompressible non-equilibrium model to model boiling flows, using a phase change model that accounts for heat and mass transfer as well as the micro-scale evaporation between the two phases [32]. While both Schmidt et al. and Kunkelmann et al. had success imple-
menting incompressible phase change models in certain boiling flows, this could become an issue in modeling cavitation damage as compressibility effects become very important during bubble collapse [3]. Some investigation would be necessary to see if these simpler non-equilibrium models could be integrated into compressible flow models.

1.2.3.2 Single Bubble Collapse Near a Solid Boundary

The motivation behind looking at the formation and collapse of a single bubble is to eventually develop a flow model that can be used to computationally model the collapse of a bubble against a solid boundary, as it relates to cavitation damage. Many researchers have focused on using some form of homogeneous flow model to look at bubble collapse near a boundary. Doihara and Takahashi were able to model bubble collapse near a boundary using a compressible axisymmetric multiphase flow model where the mass, momentum and energy conservation equations were solved ignoring the effects of viscosity, surface tension and gravity using a Cubic Interpolated Propagation based method [33]. The Tait equation of state was used for the liquid and two additional equations of state were used to calculate the sound velocity [33]. Although unable to accurately model the shock wave generated upon collapse, Doihara and Takahashi were able to model the high-speed jet that forms during bubble collapse [33]. They confirmed the experimental observations of various researchers [17, 18, 19, 21], that standoff distance is an important parameter that affects the dynamics of the jet formation [33].

Although neglecting the effects of viscosity is a common assumption [25, 33, 34, 35], Popinet and Zaleski argued that for bubbles in the range of 2 to 50 microns both thermal effects and viscous effects become important [36]. Using an finite volume method, Popinet and Zaleski were able to simulate bubble collapse near a solid boundary using the incompressible Navier-Stokes equations [36]. Their study on the effects of viscosity found that below a certain Reynolds number a jet would not appear, furthermore, they found that the distance to the wall has an impact on jet velocity [36]. While Popinet and Zaleski, as well as Doihara and Takahasi, focused on the impact of the jet when a bubble collapses near a solid surface, Johnsen and Colonius focused on a numerical method that would accurately model the shock wave generated upon collapse [25].
As previously discussed, Johnsen and Colonius used a compressible multiphase code to model bubble collapse, and assumed that the effects of viscosity, surface tension and gravity could be neglected. They were able to model both Rayleigh bubble collapse as well as shock induced collapse; their model for Rayleigh collapse included both the re-entrant jet and the shock wave [25]. They were able to model the complex pressure distribution surrounding the bubble collapse; including the shock emission from the bubble collapse, the water-hammer pressure from the jet and the maximum wall pressure [25]. These parameters, especially the maximum wall pressure, are important indicators of the impact of bubble collapse on the wall.

In addition to the field of cavitation damage, single bubble collapse near a solid boundary is also important in the simulation of underwater explosion bubbles near solid boundaries. These simulations are similar to those for cavitation bubble collapse in that they are primarily conducted using multiphase CFD models based on traditional bubble dynamics. Chan et al. used a pressure based CFD solver to model the collapse of a single bubble near a boundary [34]. Assuming that the flow was axisymmetric and that the effects of surface tension and viscosity could be ignored, as well as the effects of mass and energy transfer, Chan et al. were able to use a finite difference code to solve the unsteady, compressible two-phase Navier-Stokes equations with two additional equations of state [34]. They collapsed a bubble against both a disk and a sphere and found that the impact pressure was much higher when the bubble was collapsed above the disk instead of below it, they also found that the impact pressure from the jet was greater in the sphere [34]. They determined that the distance between the bubble collapse and the plate was an important parameter, and found that for a bubble closer to the disk the jet velocity emitted upon collapse was smaller in magnitude than for a bubble collapsing farther away from the disk [34]. The same group later modeled the formation and collapse of an underwater explosion bubble using empirical formulas to determine the initial bubble condition [35]. They were also able to produce results that were comparable to experimental data from underwater explosion bubbles near solid boundaries [35].
1.3 Objective

In order to better understand the cavitation erosion process, the formation and collapse of a single bubble will be studied. Previous research indicates that the collapse of a cavitation bubble can be modeled using compressible multiphase methods. The goal of this study is to use CFD to investigate the physics of laser-generated bubble formation and collapse. A thermal analysis of the energy profile of the laser will be undertaken using CFD to model the heat generated by a Nd:YAG laser. The collapse of a single bubble will also be analyzed using a compressible two-phase CFD solver. The dynamics of the bubble will be analyzed and comparisons will be made between the simulations and analytical solutions of single bubble formation and collapse, as well as comparisons between simulations of bubble collapse against a boundary and data collected from bubble collapse experiments.

1.4 Agenda

The fluid dynamics of cavitation bubble collapse will be analyzed using the open source CFD tool OpenFOAM. The analytical solution of the Rayleigh-Plesset equation will be found using an ODE solver in OpenFOAM. The CFD simulations of the laser-generated temperature field will also be performed in OpenFOAM, using a modified version of an existing solver. Finally a compressible two-phase flow method developed specifically for OpenFOAM by Miller et al. [37], will be used to model the collapse of a cavitation bubble. The formation and collapse of a single bubble far away from a surface will be compared both to the solution of the Rayleigh-Plesset equation as well as to experimental data of laser-generated bubble collapse. The compressible solver will also be used to model the collapse of a cavitation bubble near a solid surface. The standoff distance, the distance between the center of the bubble and the solid surface, will be varied in order to determine how the bubble dynamics are influenced by distance to the wall. The results of these simulations will then be compared with experimental results of laser-generated bubble collapse near a boundary.
Chapter 2

Methodology

2.1 Approach

In order to examine the collapse of a laser-generated cavitation bubble, the formation of the bubble will first be studied. Although a complete analysis will not be undertaken with respect to studying the formation of the bubble, a thermal analysis will be conducted of the heat generated by a laser in an actual experimental set-up. A description of the experiment will be provided, as the CFD analysis depends on the specifics of the experiment. The goal of the CFD analysis is to find the temperature distribution of the laser as it heats the water, this was accomplished using a source term that accounts for the laser heating. A CFD analysis of the collapse of a single cavitation bubble will be conducted using a two-phase compressible flow solver which will be outlined in detail. All of the CFD will be carried out using OpenFOAM, an open source CFD toolbox. A detailed summary of OpenFOAM and the numerical methods used by OpenFOAM will be given.

2.2 CFD of Laser Heating

A complete approach to modeling the formation of a laser-generated cavitation bubble would use the mass, momentum and energy equations to yield a coupled solution for the pressure, velocity and temperature fields. However, at this time the focus will be solely
on the energy equation and how the heat diffusion from the laser can be modeled using a source term. This source term takes into account the physics of the laser process and how it affects the temperature distribution of the water during and after the laser pulse. After developing a set of equations which include this source term, the source term will then be integrated into an existing solver in OpenFOAM.

2.2.1 Experimental setup of a Laser-Generated Cavitation Bubble

Using a simple setup where the laser was focused in a small tank of water, it was possible to confirm that a laser could generate a small cavitation bubble. A pulsed Nd:YAG laser with a wavelength of 532 nm, produced by frequency doubling, was used to produce the bubble. The laser has a pulse duration of 10 ns and an energy input as high as 120 mJ/pulse. A beam expander was used to first expand the laser beam, and a 200 mm lens was then used to focus the 18 mm diameter laser beam to a focal volume size of approximately 7.53 microns in diameter by roughly 83.67 microns long in the water.

![Figure 2.1: Schematic of laser-generated cavitation bubble experiment that will be computationally modeled](image)

A laser pulse of approximately 30 mJ/pulse was needed for a bubble to form with a maximum bubble radius on the order of 1 mm using the experimental setup of Figure 2.1. It is possible to change the maximum bubble size, and therefore the collapse time, by changing the amount of energy from the laser into the system.

2.2.2 Governing Equations

For a laser generated pulse the heat equation has an energy source term, $A$, that is dependent on the maximum laser intensity, the reflectivity, the absorption coefficient of
the water at the specific wavelength and the normalized spatial and temporal profiles of the laser pulse [39].

\[
\rho_c \frac{\partial T}{\partial t} = k \nabla^2 T + A
\]  

(2.1)

\[
A(x, r, t) = I_m (1 - R_w) a_w e^{-a_w x} f(r, x) q(t)
\]  

(2.2)

The source term, \(A\), and the test setup depend on the variables in \(A\); which will vary with different lasers and different parameters of the experiment. The first term is the maximum laser intensity, \(I_m\), which is defined as the laser power divided by the area of the beam. There are two constant coefficients in the source term, the reflectivity \(R_w\) and the absorption coefficient \(a_w\). They are both dependent on the wavelength of the laser (532 nm) and the medium being penetrated by the laser beam (tap water) in this experiment. The spatial profile for this laser is dependent on the beam radius, \(w\), and follows an approximate Gaussian intensity profile with the peak intensity at the center of the beam. The laser pulse can be approximated by a triangular-shaped temporal profile that is dependent on the time \(t\) and the rise time of the peak laser intensity \(\alpha\). For a laser with a Gaussian intensity profile and a triangular-shaped temporal profile, the source term can be written as follows [40, 41].

\[
A(x, r, t) = \frac{2P}{\pi w(x)^2} \left(1 - R_w\right) a_w e^{-a_w x} e^{-2\left(\frac{r}{w(x)}\right)^2} \frac{t}{a} \quad \text{for } 0 \leq t \leq a
\]  

(2.3)

\[
A(x, r, t) = \frac{2P}{\pi w(x)^2} (1 - R_w) a_w e^{-a_w x} e^{-2\left(\frac{r}{w(x)}\right)^2} \frac{2a - t}{a} \quad \text{for } a \leq t \leq 2a
\]  

(2.4)

2.2.3 Beam Diameter

In order to get an accurate profile of the laser heating it is imperative that the beam radius, sometimes referred to as the beam waist, is accurately modeled. Using conventional optics, the variation of the beam waist in the \(x\) direction can be calculated at the focal point of the laser beam using the following equation [42].
\[ w(x) = w_o \sqrt{1 + \left( \frac{x}{z_o} \right)^2} \]  

(2.5) 

When the distance \( x \) is much greater than the Rayleigh length, \( z_o \), the following approximation is used to determine the beam radius [42].

**Approximation:** \( z >> z_o \)

\[ w(x) \approx w_o \frac{x}{z_o} = \frac{\lambda x}{\pi w_o} \]  

(2.6) 

Where \( \lambda \) is the wavelength of the laser, and \( w_o \) is the waist size, or beam radius, at the focal point. The Rayleigh length \( z_o \) is calculated using the wavelength and the waist size [42].

\[ z_o = \frac{\pi w_o^2}{\lambda} \]  

(2.7) 

The waist size at the focal point, \( w_o \), is dependent on the focal lens being used. The waist size will be a function of wavelength, focal length of the lens \( f \), and the beam radius at the lens \( w_f \) [42].

\[ w_o \approx \frac{\lambda f}{\pi w_f} \]  

(2.8) 

Two additional terms which are commonly used to characterize the beam size include the confocal parameter and the focal volume. The confocal parameter is simply twice the Rayleigh length [43]:

\[ b = 2(z_o) \]  

(2.9) 

The focal volume is the volume of the liquid being heated by the laser [43].

\[ V_{focal} = \pi w_o^2 b = \frac{2\pi^2 w_o^4}{\lambda} \]  

(2.10) 

### 2.2.4 Flow Solver

The flow solver used to model laser heating was a modified version of an existing OpenFOAM solver called scalarTransportFoam. The solver scalarTransportFoam solves a
generic transport equation where $\phi$ is the transported scalar, and $u$ is the velocity [44].

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \vec{u}) - \nabla^2 (D\phi) = 0$$  \hspace{1cm} (2.11)

For the CFD of laser heating the transported scalar would be the temperature $T$, the velocity of the flow would be zero and the constant $D_\phi$ or $D_T$ would represent the thermal diffusivity of the water. Although thermal diffusivity is often denoted as $\alpha$, it will be referred to as $D_T$ to avoid confusion with the volume fraction $\alpha$. The solver was further modified to include the source term discussed in section 2.2.2, where the source term $A$ is a function of position and time. After modifying the flow solver, equation (2.11) can be written in the following manner.

$$\frac{\partial T}{\partial t} - \nabla^2 (D_T T) = \frac{A}{\rho c_p}$$  \hspace{1cm} (2.12)

After these modifications, the flow solver now solves the governing equation described in section 2.2.2.

### 2.3 Compressible Multiphase Model

A compressible two-phase approach will be used to analyze the formation and collapse of a single free cavitation bubble, as well as the collapse of a single bubble against a solid boundary. The solver that will be used is a compressible multiphase homogeneous pressure based finite volume solver created specifically for OpenFOAM by Miller et al. [37] to study underwater explosions.

#### 2.3.1 Governing Equations

The formation and collapse of a laser-generated cavitation bubble is a multiphase flow problem, with gas developing inside the bubble as it forms and collapses. For a multiphase flow the volume fraction is used as an indicator of the ratio of the phases at any point in the flow; it is defined as the fraction of the cell volume occupied by a specific
phase [27].

\[ \alpha_N = \frac{V_N}{V} \quad (2.13) \]

In a multiphase flow there is a continuity equation for each phase present in the flow; however, since the overall mass of the phases must still be conserved it is possible to write a combined continuity equation for all of the phases [27].

\[
\frac{\partial}{\partial t} \left( \sum_N \rho_N \alpha_N \right) + \frac{\partial}{\partial x_i} \left( \sum_N \rho_N \alpha_N u^i_N \right) = 0 \quad (2.14)
\]

Therefore equation (2.14) represents the continuity equation for both the liquid and gas phases, where \( \rho \) represents the density of the phase and \( \alpha \) represents the volume fraction of the phase.

The momentum equation can be similarly combined, resulting in the following equation for both the liquid and gas phases [27].

\[
\frac{\partial}{\partial t} \left( \sum_N \rho_N \alpha_N u^i_N u^i_N \right) + \frac{\partial}{\partial x_i} \left( \sum_N \rho_N \alpha_N u^i_N u^i_N \right) = p g_k + \frac{\partial p}{\partial x_k} + \frac{\partial \sigma^D_{Cki}}{\partial x_i} \quad (2.15)
\]

Where \( g_k \) is the gravitational acceleration, \( p \) is the pressure and \( \sigma^D_{Cki} \) is the deviatoric stress tensor.

### 2.3.2 Flow Solver

Although a complete set of transport equations could have been solved for each phase, in order to reduce the complexity of the solver a homogeneous flow model was used instead. The homogeneous model assumes that the velocity and the temperature of the two phases are approximately equal at the interface [27, 37]. This assumption leads to a reduction in governing equations, and the conservation of mass and momentum equations reduce to those for single phase flow [27, 37]. The conservation of mass and momentum for a compressible homogeneous flow are given as follows [37].

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2.16)
\]
\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \text{div}(\rho \mathbf{u} \otimes \mathbf{u}) - (\text{grad} \rho \mathbf{u}) \text{grad} \mu_{\text{eff}} - \mu_{\text{eff}} \Delta \mathbf{u} = -\text{grad} p_d - (\text{grad} \rho) \mathbf{g} \cdot \mathbf{x} \quad (2.17)
\]

Where \( \mu_{\text{eff}} \) is the effective viscosity, \( \mu_{\text{eff}} = 4(\alpha_1 \mu_1 + \alpha_2 \mu_2)/3 \), and \( p_d \) is the piezometric pressure, \( p_d = p - \rho \mathbf{g} \cdot \mathbf{x} \) [37]. The density however, will vary for each phase, it is for this reason that a mixture density is introduced [37].

\[
\rho = \alpha_1 \rho_1 + \alpha_2 \rho_2 \quad (2.18)
\]

Two equations of state will be used to calculate the density of each phase and will then be used, along with the volume fractions of each phase, to find the mixture density \( \rho \). For a two-phase flow the volume fractions must sum to one [37].

\[
\alpha_1 + \alpha_2 = 1 \quad (2.19)
\]

After significant manipulation equation (2.16) and equation (2.19) can be rewritten as a statement of total mass balance, a volume fraction equation and a pressure equation [37].

\[
\left( \frac{\alpha_1}{\rho_1} \frac{\partial \rho_1}{\partial p} + \frac{\alpha_2}{\rho_2} \frac{\partial \rho_2}{\partial p} \right) (\dot{p} + \mathbf{u} \cdot \text{grad} p) + \text{div} \mathbf{u} = 0 \quad (2.20)
\]

\[
\dot{\alpha}_1 + \text{div} \alpha_1 \mathbf{u} - \alpha_1 \text{div} \mathbf{u} = \alpha_1 \alpha_2 \left( \frac{\partial \rho_1}{\partial p} \rho_2 - \frac{\partial \rho_2}{\partial p} \rho_1 }{\alpha_1 \psi_1 \rho_2 + \alpha_2 \frac{\partial \rho_2}{\partial p} \rho_1} \right) \text{div} \mathbf{u} \quad (2.21)
\]

Where \( \psi = 1/c^2 \), where \( c \) is the speed of sound.

The three governing equations that are solved are equations (2.17), (2.20), and (2.21). These governing equations are used to solve for \( \mathbf{u}, p \) and \( \alpha_1 \) respectively, while the remaining three unknowns, \( \alpha_2, \rho_1 \) and \( \rho_2 \), are solved using equation (2.19) to calculate \( \alpha_2 \) and two equations of state to find \( \rho_1 \) and \( \rho_2 \). The equation of state for the gas phase is based on the assumption that the vapor inside of the bubble behaves as an ideal gas and only undergoes isentropic processes, resulting in the following equation of state [37].

\[
\rho = \left( \frac{p}{a_c} \right)^{\frac{1}{\gamma}} \quad (2.22)
\]
Where $p$ is the pressure of the gas, $a_c$ is the adiabatic constant and $\kappa$ is the ratio of specific heats.

The equation of state for the liquid phase is derived from the speed of sound in a medium, assuming that the sound speed is constant and the process is isentropic [37].

$$\rho = \rho_0 + \psi(p - p_0)$$

(2.23)

Where $\rho_0$ and $p_0$ are the reference density and reference pressure, and $c$ is the speed of sound. The momentum, pressure and volume fractions are all linearized and the complete set of equations is solved using a cell-centered, co-located finite volume method [37]. For a more detailed description of the implementation of the method, including the discretization of the governing equations and a complete description of the solution algorithm see Miller et al. [37]. All of the compressible multiphase CFD analysis in this thesis uses this OpenFOAM solver.

### 2.4 Numerical Methods

Computational fluid dynamics (CFD) is useful when partial differential equations cannot be solved analytically. An approximate solution to the partial differential equations can be found using discretized forms of the PDEs, which can then be solved algebraically on a computer [45]. There are three primary discretization methods; the finite difference method, the finite volume method and the finite element method [45]. The main advantage of a finite volume method is that it can be used on any type of grid and for complex geometries [45], it is the method used in OpenFOAM [44]. The choice of coordinate system and grid is another important decision in numerical modeling. OpenFOAM uses a three-dimensional right-handed Cartesian coordinate system, and is capable of handling both structured and unstructured grids [46].

There are many different discretization schemes applicable to finite volume methods, including first-order schemes such as Upwind Interpolation (UDS), and second-order schemes such as Linear Interpolation (CDS) and Quadratic Upwind Interpolation (QUICK) [45]. There are also numerous scheme variations and correction factors avail-
able, many of which are included in OpenFOAM’s library [44]. For unsteady problems the solution is time dependent, and therefore the time derivatives must be discretized as well. For explicit methods, such as the Explicit Euler Method and the Leapfrog Method, the solution in time is dependent on the solution at the previous time step [45]. For implicit methods, such as the Implicit Euler Methods and the Crank-Nicolson Method, the solution depends on both the current and future state of the system, and although harder to implement they offer greater numerical stability [45]. Besides stability, it is also important that the solution is convergent, that the governing equations are conserved and that the solution is properly bounded [45]. Although numerical solutions are by definition approximate solutions, choosing an appropriate coordinate system, grid, discretization method and schemes can yield an excellent approximation of the solution.

2.5 OpenFOAM

OpenFOAM, which stands for Open Source Field Operation and Manipulation, is a collection of C++ libraries created as an open source CFD tool [47]. OpenFOAM includes numerous solvers and utilities, all of which can be used and modified by the user [47]. The standard solver scalarTransportFoam was modified for the thermal analysis of the formation of a laser-generated cavitation bubble, while the compressible flow solver used for the bubble collapse analysis was created within the OpenFOAM framework. One of the benefits of using OpenFOAM is that it is possible to create new solvers using existing libraries, it is also possible to make or modify a code for a specific need [44]. Therefore, it is simpler than writing a complete CFD code, but more flexible than using software where the code cannot be modified.

In addition to numerous solvers, OpenFOAM also contains its own mesh generation tools. Of the possible meshing tools the simplest is blockMesh, which can be used for a variety of simple geometries [44]. In blockMesh the user specifies the vertices, the number of cells, the grading of the cells and the type of boundary conditions [47]. By default OpenFOAM solves a system of equations in three dimensions, so for two dimensional or axisymmetric solutions the special boundary conditions “empty” and “wedge”
must be used to denote that the solution is not three dimensional [47]. The boundary conditions, as well as the initial conditions, are specified in the sub-directory 0, while the fluid properties and the mesh generation files are contained in the sub-directory constant [47]. An additional sub-directory system, contains the files that control the numerical parameters of the simulation; including the time step, the run-time parameters, the numerical schemes, the equation solvers and their tolerances [47]. OpenFOAM is compatible with many other meshing tools, such as Pointwise and Gridgen, as well as with the post-processing program ParaView.

2.6 Summary

In order to model the collapse of a laser-generated cavitation bubble, as well as the temperature profile generated by the laser, computational solutions will be found using OpenFOAM. The first step was to determine the governing equations, or the mathematical models, necessary for both cases. After determining what the appropriate equations were, a discretization method was chosen to find an approximate solution to those equations. The finite volume method and a three dimensional Cartesian coordinate system were chosen, as they are the method and coordinate system used in OpenFOAM. The meshes will be generated in blockMesh, or Pointwise, while the boundary and initial conditions will be set in the OpenFOAM directory. The discretization schemes, the fluid properties and the run-time parameters will also be chosen in OpenFOAM. After ensuring that the correct parameters have been chosen, the numerical solution will be determined using the designated flow solver in OpenFOAM. A thermal analysis of the energy generated by the laser will be conducted using a modified version of the standard OpenFOAM solver scalarTransportFoam, while collapse of a single laser-generated cavitation bubble will be modeled using a compressible multiphase solver. Comparisons will be made between the compressible multiphase solution, the analytical solution of the Rayleigh-Plesset equation and experimental data. The multiphase solver will then be used to model the collapse of a single cavitation bubble against a solid boundary.
Cavitation bubbles were investigated as early as 1917 by Lord Rayleigh, who subsequently derived what is now known today as the Rayleigh model [3]. The Rayleigh model is a simplified form of the Navier-Stokes equation in the radial direction assuming that a spherical bubble forms and collapses in an infinite medium [1]. The Rayleigh-Plesset equation is the Rayleigh model with the inclusion of terms accounting for surface tension, liquid viscosity and the gas contents of the bubble [1, 5]. Despite its simplicity, the Rayleigh-Plesset model is a good predictor of most spherical bubble oscillations [5], and is therefore a crucial part of this analysis.

3.1 Approach

The Rayleigh-Plesset equation, although a simplified model for bubble formation and collapse, can still offer useful information. Given a set of initial conditions, it can provide an estimate of the maximum bubble radius and the time of the bubble collapse. Or if those conditions are known, it can provide an estimate of the initial pressure of the gas inside the bubble, or of the initial bubble size. Since there are many assumptions made in the derivation of the Rayleigh-Plesset equation, these assumptions will be covered in detail as they are important to an accurate analysis. The time scales of the Rayleigh-Plesset equation will also be discussed and analyzed; including the bubble collapse time. The Rayleigh-Plesset equation will then be used to analyze the oscillations of a cavitation
bubble of approximately the same size as a laser-generated bubble.

3.2 Rayleigh-Plesset equation

The derivation of the Rayleigh-Plesset equation starts with the assumption that there is a perfectly spherical bubble in an infinite fluid, where the liquid far from the bubble is assumed to be at a certain pressure and temperature, \( p_\infty(t) \) and \( T_\infty \) \([1]\). The pressure far away from the bubble is changing with time, while the temperature far away from the bubble is assumed to be constant \([1]\). It is assumed that the fluid is incompressible and has constant properties, and that the contents of the bubble are homogeneous \([1]\). Further assumptions are that the flow is only occurring in the radial direction, and that there is zero mass transfer across the liquid-vapor interface \([1]\).

3.2.1 Governing Equation

The first governing equation is the continuity equation, where for spherical coordinates \([48]\):

\[
\frac{\partial p}{\partial t} + \frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 \rho u_r \right) + \frac{1}{rsin\theta} \frac{\partial}{\partial \theta} \left( \rho u_\theta sin \theta \right) + \frac{1}{rsin\theta} \frac{\partial}{\partial \phi} \left( \rho u_\phi \right) = 0 \quad (3.1)
\]

The second governing equation is the momentum equation, where for spherical coordinates \([48]\):

\[
\frac{\partial u_r}{\partial t} + (\vec{u} \cdot \nabla) u_r - \frac{u_\theta^2 + u_\phi^2}{r} = - \frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left[ \nabla^2 u_r - \frac{2u_r}{r^2} - \frac{2u_r}{r^2 sin \theta} \frac{\partial (u_\theta sin \theta)}{\partial \theta} - \frac{2}{r^2 sin \theta} \frac{\partial u_\phi}{\partial \phi} \right] \quad (3.2)
\]

3.2.1.1 Derivation of the Rayleigh-Plesset equation

Assuming that the flow is incompressible and occurs only in the \( r \) direction, due to spherical symmetry, the continuity equation (3.1) then reduces to:

\[
\frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 \rho u_r \right) = 0 \quad (3.3)
\]

Integrating (3.3) yields the following equation:
\[ u_r(r, t) = \frac{C_1(t)}{r^2} \]  
(3.4)

The following two boundary conditions are then applied, since it has been assumed that there is zero mass transport across the interface \([1]\):

\[ u_r(R, t) = \frac{dR}{dt} \]  
(3.5)

\[ r = R(t) \]  
(3.6)

The continuity equation then becomes:

\[ u_r(r, t) = \frac{dR}{dt} \frac{R^2}{r^2} \]  
(3.7)

The momentum equation (3.2) reduces to:

\[ \frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} = -\frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left[ \frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 \frac{\partial u_r}{\partial r} \right) - \frac{2u_r}{r^2} \right] \]  
(3.8)

Now substitute (3.7) into (3.8) and simplify:

\[ \frac{1}{r^2} \frac{d}{dt} \left[ \left( 1 - \frac{\rho_v}{\rho_L} \right) R^2 \frac{dR}{dt} \right] - \frac{2}{r^5} \left[ \left( 1 - \frac{\rho_v}{\rho_L} \right) R^2 \frac{dR}{dt} \right] = -\frac{1}{\rho_L} \frac{dp}{dr} \]  
(3.9)

Which reduces to:

\[ \frac{1}{r^2} \frac{d}{dt} \left( \frac{dR}{dt} R^2 \right) - \frac{2}{r^5} \left( \frac{dR}{dt} R^2 \right)^2 = -\frac{1}{\rho_L} \frac{dp}{dr} \]  
(3.10)

The next step is to integrate and recall that as \( r \to \infty, p \to \infty \) \([1]\).

\[ \frac{p(r, t) - p_\infty(t)}{\rho_L} = \frac{1}{r} \frac{d}{dt} \left[ \left( 1 - \frac{\rho_v}{\rho_L} \right) R^2 \frac{dR}{dt} \right] - \frac{1}{2r^4} \left[ \left( 1 - \frac{\rho_v}{\rho_L} \right) R^2 \frac{dR}{dt} \right]^2 \]  
(3.11)

The balance of the normal forces at the interface yields the following boundary condition; where \( \sigma_{rr} \) is the stress tensor, \( p_B \) is the pressure inside the bubble and \( S \) is the surface tension \([1, 3]\).

\[ \text{at } r = R(t) : \quad (\sigma_{rr})_{r=R} + p_B - \frac{2S}{R} = 0 \]  
(3.12)
Recall that $\sigma_{rr} = -p + 2\mu \frac{\partial u}{\partial r}$, (3.12) then becomes [1]:

$$p_B - (p)_{r=R} = \frac{4\mu L}{R} \frac{dR}{dt} - \frac{2S}{R}$$  \hspace{1cm} (3.13)

Now consider that at $r = R$, (3.7) and (3.13) can be substituted into (3.11), which is then simplified and becomes the Rayleigh-Plesset equation [1].

$$\frac{p_B(t) - p_\infty(t)}{\rho_L} = R \frac{d^2R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 + \frac{4\nu_L}{R} \frac{dR}{dt} + \frac{2S}{\rho_L R}$$  \hspace{1cm} (3.14)

This form of the Rayleigh-Plesset equation does not account for the contents of the bubble; it can be assumed that the bubble contains a certain amount of noncondensable gas whose partial pressure is $p_{G_0}$ at an initial radius of $R_0$ and a uniform temperature $T_\infty$ [1]. A further assumption that can be made is that there are no significant thermal effects in the bubble behavior, this means that $T_\infty$ is indeed uniform and that the change in vapor pressure as time increases is negligible [1]. The pressure inside the bubble is usually then calculated using either a polytropic relationship [1] or a van der Waals law [5]. After including these assumptions and the polytropic relation into (3.14), the Rayleigh-Plesset equation can be written in the following form [1].

$$\frac{p_v(T_\infty) - p_\infty(t)}{\rho_L} + \frac{p_{G_0}}{\rho_L} \left( \frac{R_0}{R} \right)^{3\kappa} = R \ddot{R} + \frac{3}{2} \left( \dot{R} \right)^2 + \frac{4\nu_L}{\rho_L} \frac{\dot{R}}{R} + \frac{2S}{\rho_L R}$$  \hspace{1cm} (3.15)

Many of the assumptions that were made in the derivation of the Rayleigh-Plesset equation are valid assumptions for a simplified model of the growth and collapse of a laser-induced cavitation bubble. Bubbles generated by lasers are nearly spherical [5], and for the experimental setup of a single bubble generated in a large tank of water the surrounding fluid is much larger than the bubble, therefore the assumption that the bubble is generated in an infinite fluid is justified. Furthermore, even though the generation and collapse of the laser-generated cavitation bubble significantly changes the properties of the nearby fluid, since the formation and collapse is occurring at a micro-scale level it can be argued that the water far away from the collapse will not change in either pressure or temperature. As previously discussed the compressibility of the liquid will become important during collapse; however, an incompressible model should give a
very good representation of the formation of the bubble and an approximation of the bubble collapse.

In the context of the natural formation of a cavitation bubble, the assumption that there is no significant thermal effects in the bubble growth is a fairly good approximation, as the change in the temperature inside the bubble is very small [3]. For a laser-generated cavitation bubble however, the reverse is true, the temperature of the water is rapidly changing; in fact it is the temperature difference that causes the bubble to form. As previously discussed a good model of the formation of a laser-generated cavitation bubble should accurately capture the heat and mass transfer across the liquid-vapor interface, something not accounted for in the Rayleigh-Plesset equation. Not much is known about the gas contents inside a laser-generated cavitation bubble [26], however, for the purposes of this simplified model it was assumed that the gas was noncondensable and could be modeled with a polytropic relationship. Despite the limitations of the Rayleigh-Plesset model, it will be used as a simplified model to verify a more complex model of bubble collapse.

3.2.2 Time Scales

Since bubble collapse is a time-dependent problem, it is of interest to analyze the time scales of the Rayleigh-Plesset equation. Recall that the Rayleigh-Plesset equation is dependent on position $R$, as well as the first and second derivatives of $R$, $\dot{R}$ and $\ddot{R}$, which represent the velocity and acceleration of the surface of the bubble respectively.

Perhaps the most important time scale is the time required for total collapse, sometimes called the Rayleigh time, which is derived from (3.15) when the effects of viscosity, noncondensable gas and surface tension are ignored [1, 3, 49].

$$\tau_c \approx 0.915 R_{max} \sqrt{\frac{\rho_L}{p_\infty - p_v}}$$

(3.16)

This relationship is important because it gives a very good approximation between the maximum bubble size and the collapse time of the bubble. Since the fluid properties of bubble collapse in a free field are typically approximated as constant, a change in the collapse time is only noticeable when the maximum radius changes.
Examining the nondimensional form of the Rayleigh-Plesset equation yields other time scales of interest. In nondimensionalizing any equation a characteristic length and time scale must be chosen; for this equation the initial bubble radius $R_0$, the collapse time $\tau_c$ and the pressure difference $P = p_\infty - p_v$ have been chosen to represent the characteristic scales of radius, time and pressure respectively [3]. From this nondimensional form of the Rayleigh-Plesset equation three time scales of interest can be found: the pressure time $\tau_p$, the viscous time $\tau_\nu$ and the surface tension time $\tau_s$ which are defined as follows [3]:

$$\tau_p = R_0 \sqrt{\frac{p_L}{P}} \approx \tau_C$$

$$\tau_\nu = \frac{R_0^2}{4 \nu L}$$

$$\tau_s = R_0 \sqrt{\frac{p_L R_0}{2S}}$$

Where the Reynolds number is a nondimensional parameter given by [3]:

$$Re = \frac{\tau_\nu}{\tau_c}$$

These time scales are important for two reasons; the first is that there is now a simple way to relate the time scales and the Reynolds Number, the second is that when solving the Rayleigh-Plesset equation computationally the time step should be smaller than the above time scales [3].

To see which time scale is typically the shortest each time scale was computed using typical values for a laser-generated cavitation bubble in water assuming that all the values remain constant despite changes in temperature and pressure: $\rho_L = 1000 \text{ kg/m}^3$, $p_\infty = 0.1 \text{ MPa}$, $p_v = 2.33 \text{ kPa}$, $\nu_L = 1 \times 10^{-6} \text{ m}^2/\text{s}$, and $S = 0.0725 \text{ N/m}$.  

Table 3.1 illustrates the importance of the bubble size on the time scales. For a cavitation bubble with an initial radius of 1 mm, $\tau_p$ is smaller than $\tau_S$ and significantly smaller than $\tau_\nu$. For a cavitation bubble with an initial radius of 1 µm, $\tau_S$ is smaller than
Table 3.1: Time scales of the Rayleigh-Plesset equation for different $R_0$

<table>
<thead>
<tr>
<th>$R_0$ (m)</th>
<th>$\tau_p$ (s)</th>
<th>$\tau_\nu$ (s)</th>
<th>$\tau_S$ (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1 \times 10^{-6}$</td>
<td>$1.01 \times 10^{-7}$ s</td>
<td>$2.50 \times 10^{-7}$ s</td>
<td>$8.30 \times 10^{-8}$ s</td>
</tr>
<tr>
<td>$1 \times 10^{-5}$</td>
<td>$1.01 \times 10^{-6}$ s</td>
<td>$2.50 \times 10^{-5}$ s</td>
<td>$2.63 \times 10^{-6}$ s</td>
</tr>
<tr>
<td>$1 \times 10^{-4}$</td>
<td>$1.01 \times 10^{-5}$ s</td>
<td>$2.50 \times 10^{-3}$ s</td>
<td>$8.30 \times 10^{-5}$ s</td>
</tr>
<tr>
<td>$1 \times 10^{-3}$</td>
<td>$1.01 \times 10^{-4}$ s</td>
<td>$2.50 \times 10^{-1}$ s</td>
<td>$2.63 \times 10^{-3}$ s</td>
</tr>
</tbody>
</table>

$\tau_p$ and $\tau_\nu$ which are approximately the same size. As will be discussed in section 3.5.3, there is still much debate about the size of the minimum radius of a laser-generated cavitation bubble. The ratio between the maximum bubble radius and the minimum bubble radius has been experimentally observed to be anywhere between 20 and 70 [17, 22, 26]. For a bubble with a maximum radius of 1 mm, the initial radius would therefore be on the order of $1 \times 10^{-5}$ m to $1 \times 10^{-4}$ m.

3.3 Numerical Methods

OpenFOAM includes an ordinary differential equation, or ODE solver, that was used to solve the Rayleigh-Plesset equation. The Rayleigh-Plesset equation (3.15) was implemented into the ODE solver using a form of (3.17) to determine an appropriate time step. The ODE solver requires that the Rayleigh-Plesset equation, a second order ODE, is separated into two first order ODEs. The solver also needed the Jacobian of the two equations in order to calculate a solution. The ODE solver in OpenFOAM includes a Runge-Kutta scheme [47], which was used to solve the ODE in this analysis.

3.4 Fluid Properties

Table 3.2 shows the fluid parameters that will be used in the solution of the Rayleigh-Plesset equation. All of the fluid parameters used were determined [50] assuming that the bubble forms in water at standard atmospheric pressure and temperature.

Table 3.2: Fluid parameters for the solution of the Rayleigh-Plesset equation

<table>
<thead>
<tr>
<th>$\rho_L$ (kg/m$^3$)</th>
<th>$p_\infty$ (Pa)</th>
<th>$p_\nu$ (Pa)</th>
<th>$\nu$ (m$^2$/s)</th>
<th>$S$ (N/m)</th>
<th>$\kappa$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
<td>101325</td>
<td>$2.33 \times 10^3$</td>
<td>$1 \times 10^{-6}$</td>
<td>0.072</td>
<td>1.4</td>
</tr>
</tbody>
</table>
3.5 Bubble Dynamics of Single Bubbles using the Rayleigh-Plesset equation

3.5.1 Single Bubble Dynamics of the Rayleigh-Plesset equation

In order to verify that the numerical method described in section 3.3 correctly solves the Rayleigh-Plesset equation, a simple test case will be examined. For a single bubble forming and collapsing in a free field the bubble will achieve a maximum radius, $R_{\text{max}}$, collapse, and then achieve a minimum radius at a certain collapse time, $\tau_c$. For a bubble with an initial radius of 10 $\mu$m, an initial gas pressure of 40 GPa and the fluid properties outlined in section 3.4, a solution to the Rayleigh-Plesset equation was determined using the ODE solver in OpenFOAM.

![Figure 3.1: $R/R_{\text{max}}$ versus $t/\tau_c$ calculated using the Rayleigh-Plesset equation](image)

Figure 3.1 shows the radius of the bubble normalized by the maximum radius versus the solution time normalized by the collapse time of the bubble; where only one bubble oscillation is shown. This figure confirms that the ODE solver is solving the Rayleigh-Plesset equation appropriately; the normalized radius begins very close to zero and gradually approaches a value of one when $R = R_{\text{max}}$. Similarly, after the first collapse the bubble reaches a minimum radius when $t = \tau_c$, as expected. Furthermore, the spherical shape of the bubble shown in Figure 3.1 is indicative of the bubble formation.
and collapse expected for the solution of the Rayleigh-Plesset equation.

### 3.5.2 Single Cavitation Bubbles

This section will focus on the solution of the Rayleigh-Plesset equation for a typical cavitation bubble. Initially the initial radius of the bubble will always be assumed to be 1 mm for simplicity, although cavitation bubbles typically have a smaller initial radius as will be discussed in section 3.5.3.

#### 3.5.2.1 Effect of Pressure Ratio on Bubble Dynamics

Using the fluid parameters described in section 3.4, the Rayleigh-Plesset equation was solved for a bubble of initial radius 1 mm at atmospheric pressure conditions. The pressure ratio, the initial gas inside of the bubble divided by the pressure far away from the bubble, was varied holding \( p_\infty \) constant. As can be seen below in Table 3.3, when \( p_{G_0} \) was increased both the maximum radius and the collapse time increased; more than doubling as the pressure ratio is increased by a factor of ten.

<table>
<thead>
<tr>
<th>( R_0 ) (mm)</th>
<th>( R_{max} ) (mm)</th>
<th>( p_{G_0} ) (Pa)</th>
<th>( \tau_c ) (( \mu )s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>2.65</td>
<td>( 1.0 \times 10^6 )</td>
<td>585</td>
</tr>
<tr>
<td>1.0</td>
<td>6.11</td>
<td>( 1.0 \times 10^7 )</td>
<td>1200</td>
</tr>
<tr>
<td>1.0</td>
<td>13.5</td>
<td>( 1.0 \times 10^8 )</td>
<td>2548</td>
</tr>
</tbody>
</table>

It is expected that increasing the initial gas pressure inside of the bubble would increase the maximum radius, this in turn would lead to an increase in the collapse time as indicated in equation (3.16). Figure 3.2 illustrates the difference in the growth and collapse of a bubble according to the Rayleigh-Plesset equation for both a pressure ratio of 100 and a pressure ratio of 1000. As previously noted the pressure ratio must increase by a factor of ten before the maximum radius increases by a factor of two. Therefore in order to achieve a maximum radius much larger than the initial radius an extremely large pressure ratio is needed, as will be discussed further in section 3.5.3.
Figure 3.2: Radius calculated using the Rayleigh-Plesset equation versus time for different pressure ratios

Figure 3.3 is a graph of the nondimensional radius versus the nondimensional time at different pressure ratios. The nondimensional radius is the radius of the bubble divided by the initial radius of the bubble and the nondimensional time is the time divided by the collapse time of the bubble. As anticipated higher pressure ratios lead to larger bubble growth.

Figure 3.3: Nondimensional radius calculated using the Rayleigh-Plesset equation versus nondimensional time for different pressure ratios

Besides varying the pressure ratio, the solution to the Rayleigh-Plesset equation will
change if the fluid parameters or the initial radius is changed.

3.5.2.2 Effect of Initial Conditions on Bubble Dynamics

For a cavitation bubble forming and collapsing in a tank of water the fluid parameters in the Rayleigh-Plesset equation will not change much. The density of the water, the vapor pressure of the water, the liquid viscosity and the surface tension of the water are all dependent on the temperature of the water and will not change significantly for a problem that is occurring on such a short time scale. Changing any of these parameters slightly will not significantly affect the solution. In fact the viscosity and surface tension terms are so insignificant that for a bubble of this size they can be neglected. For this problem the only initial conditions that have any discernible affect on the solution are the initial radius and the initial pressure of the vapor inside of the bubble. The effect of the pressure ratio has already been discussed in detail, while changing the initial radius will only result in a corresponding change in the maximum bubble radius.

3.5.3 Single Free Cavitation Bubble

For laser-generated cavitation bubbles there is a specific laser energy range that results in the formation of only one bubble, additional energy will result in the creation of multiple bubbles. The size of the bubble is dependent on the type of laser used, the optical set-up, and the amount of energy outputted by the laser. Bubbles generated with a Nd:YAG laser typically have a maximum radius on the order of 0.1 to 1 mm [17, 19, 22, 26]. The maximum radius is generally estimated from images taken using a high-speed camera, or calculated from the collapse time recorded using a hydrophone [17, 19, 26]. The values, however, for the minimum radius of the bubble are much harder to calculate experimentally because of the limitations of the optical system and the exposure time of the camera [26]. There is significant variation in the literature of an acceptable compression ratio, \( R_{\text{max}}/R_0 \). Vogel et al. experimentally calculated a minimum radius of 50 \( \mu \text{m} \) and a maximum radius of 3.5 mm [22], yielding a compression ratio of 70. Philipp et al. reported a maximum radius of 1.45 mm and a minimum radius of 70 \( \mu \text{m} \) [17], a much smaller compression ratio. While Muller et al. reported a maximum radius
of 747 µm and a minimum radius of 12 µm [26], yielding a compression ratio of roughly 62. For the purpose of this analysis of the Rayleigh-Plesset equation, and comparison of the initial pressure conditions necessary to produce various bubble sizes, compression ratios of 20, 70 and 100 were examined.

Using the fluid parameters described in section 3.4, the Rayleigh-Plesset equation was solved for three different compression ratios, shown below in Table 3.4. For simplicity the maximum radius was assumed to be 1 mm and the initial radius was varied accordingly.

<table>
<thead>
<tr>
<th>( R_0 ) (µm)</th>
<th>( R_{\text{max}} ) (mm)</th>
<th>( p_{\text{G0}} ) (Pa)</th>
<th>( \tau_c ) (µs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.0</td>
<td>1.0</td>
<td>( 4.0 \times 10^{10} )</td>
<td>184.4</td>
</tr>
<tr>
<td>14.3</td>
<td>1.0</td>
<td>( 1.4 \times 10^{10} )</td>
<td>185.9</td>
</tr>
<tr>
<td>50.0</td>
<td>1.0</td>
<td>( 3.3 \times 10^{8} )</td>
<td>187.0</td>
</tr>
</tbody>
</table>

From Table 3.4 it can be seen that larger compression ratios result in much higher initial pressures inside the bubble. It is interesting that increasing the initial radius from 10 µm to 50 µm results in a drop of almost two orders of magnitude in the initial gas pressure. In fact the relationship between the pressure ratio and the compression ratio is in no way linear, as can be see below in Figure 3.4, where \( p_\infty \) and \( R_0 \) are held constant.

![Figure 3.4](image)

Figure 3.4: The compression ratio versus the pressure ratio for constant \( R_0 \) and \( p_\infty \)

As expected from equation (3.16) the collapse time of the bubble is dependent pri-
marily on the maximum radius of the bubble, with the minimum radius of the bubble having little to no affect on the collapse time. In the calculations of the Rayleigh-Plesset equation it is really only the initial gas pressure that is influenced by changing the initial radius, as can be seen in Table 3.4 and Figure 3.5.

![Figure 3.5: Nondimensional time versus nondimensional radius for various compression ratios](image)

### 3.6 Discussion

While the Rayleigh model is a simplified model of bubble behavior, it is a useful model because it gives a relationship between the initial pressure inside the bubble and the radius of the bubble. The Rayleigh model does assume incompressible, one dimensional flow. Therefore while the Rayleigh model may give a good approximation of the bubble radius as it first expands, as the bubble collapses compressibility effects will occur and the model will not give as good of a solution. In summary, the Rayleigh-Plesset equation should give a good prediction of the bubble’s maximum radius and collapse time. It is when the bubble starts to collapse that the Rayleigh-Plesset equation will start to overpredict the radius of the bubble. After the first oscillation of the bubble, the Rayleigh-Plesset model is no longer appropriate for modeling cavitation bubble collapse as the assumptions of the Rayleigh-Plesset equation are no longer valid.
Chapter 4

Laser-Generated Heat Source in Heat Conduction Equation

While previous research has focused on simulating the collapse of a cavitation bubble, modeling the formation of a laser-generated cavitation bubble has not yet been attempted using a thermal analysis of the heat produced by the laser. Research has been done however, regarding the heat diffusion caused by lasers in surfaces. A thermal model for laser heating is important for pulsed laser ablation processes, such as welding and drilling, where material is removed from either a solid or a liquid using a laser [51]. Previous research has focused on laser ablation of solids, liquids and polymers and has included both analytical and computational models [51]. The laser heating process is also important for laser-related medical applications, such as those used in dermatology [40] and microsurgery within the eye [23]. Since the process of laser heating in water is the same as laser heating in solids and various other liquids, it is possible to use the same methods to determine the heat diffusion caused by the laser as it generates a cavitation bubble.

4.1 Approach

Using the computational flow solver described in section 2.2.4, an analysis will be made of the temperature distribution in water caused by a laser. The focus of this analysis
is on solving the energy equation with a source term that accounts for the heat diffusion from the laser beam. The source term accounts for the type of laser used, the optical parameters of the experiment and the fluid properties of the medium that the beam is penetrating. After developing a set of governing equations, which included the laser-heating source term, these equations were integrated into an existing solver in OpenFOAM. An axisymmetric mesh was generated in BlockMesh, and a temperature distribution was determined for the laser-generated heat source.

4.2 Governing Equations

The CFD solver yields an approximate solution for the heat equation with a laser heating source term for an incompressible fluid where the velocity field is zero everywhere. A more detailed analysis of the governing equations can be found in section 2.2.2.

\[
\frac{\partial T}{\partial t} = D_T \nabla^2 T + \frac{A}{\rho c_p} \quad (4.1)
\]

Where \(A\) is the source term that accounts for the laser heating. The source term is dependent on several laser parameters including the maximum laser intensity \(I_m\), the beam radius \(w\) and the rise time of the peak laser intensity \(a\). The source term is also dependent on the properties of the water, including the reflectivity \(R_w\) and the absorption coefficient \(a_w\).

\[
A = \frac{2P}{\pi w^2} (1 - R_w) a_w e^{-a_w x} e^{-2(\frac{x}{w})^2} \frac{t}{a} \quad \text{for} \ 0 \leq t \leq a \quad (4.2)
\]

\[
A = \frac{2P}{\pi w^2} (1 - R_w) a_w e^{-a_w x} e^{-2(\frac{x}{w})^2} \frac{2a - t}{a} \quad \text{for} \ a \leq t \leq 2a \quad (4.3)
\]

4.2.1 Beam Diameter

As previously discussed in section 2.2.3, it is important that the calculated beam radius of the laser, \(w\), accurately represents the beam profile of a Gaussian laser. The beam radius was calculated using conventional optics, where the beam radius is dependent on
the wavelength of the laser $\lambda$, the Rayleigh length $z_o$, and the waist size of the beam $w_o$, as outlined in section 2.2.3. The diameter of a Gaussian laser beam has a distinct shape near the focal volume of the laser, Figure 4.1a shows the calculated beam diameter of the laser while Figure 4.1b shows the calculated beam diameter near the focal volume. The beam diameter was calculated as described in section 2.2.3 and implemented into the flow solver described in section 2.2.4.

![Figure 4.1: Calculated beam diameter of the laser: where only positive $w(z)$ is included in the CFD simulation](image)

### 4.3 Mesh Generation

The mesh for this computational simulation was generated in BlockMesh. Since the goal of the computational work is to model the formation and collapse of a laser-generated cavitation bubble, the mesh was created with the same dimensions as the experiment. By definition OpenFOAM solves all problems in three-dimensions; however, certain boundary conditions can be specified that reduce the problem to a one-dimensional or two-dimensional solution [47]. For this problem a two-dimensional axisymmetric solution was desired. A wedge mesh is a mesh constructed in the shape of a wedge having an angle less than or equal to five degrees [47]. For this mesh an angle $\theta$ of 5 degrees was chosen. The boundary condition at $r = 0$ is a symmetry plane; therefore the simulation
only models half of the laser beam. Figures 4.2 and 4.3 show the dimensions of the wedge mesh, where the mesh is a wedge section of a cylinder. The length of the mesh in the $x$ direction is much longer than the height of the mesh in the $y$ direction, making visualization of the mesh difficult.

![Figure 4.2: Axisymmetric wedge mesh for CFD of laser heating](image)

It is important that the area around the focal volume of the laser, located at the center of the mesh, is refined enough that it can accurately model the temperature gradients in that region. Since certain areas of the mesh need to be very refined, while other areas of the mesh can be less refined, the wedge is separated accordingly. In order to minimize the amount of necessary cells the mesh was divided into ten different sections, or blocks, expansion ratios were used to create a smooth transition between the different blocks. Figure 4.3 shows the division of the mesh by blocks while Table 4.1 shows the number of cells in each block.

![Figure 4.3: Block numbering of the wedge mesh for CFD of laser heating](image)
Table 4.1: Number of cells in axisymmetric wedge mesh used in CFD of laser heating

<table>
<thead>
<tr>
<th>Block Number</th>
<th>$N_x$</th>
<th>$N_y$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>1</td>
<td>250</td>
<td>200</td>
</tr>
<tr>
<td>2</td>
<td>250</td>
<td>200</td>
</tr>
<tr>
<td>3</td>
<td>250</td>
<td>200</td>
</tr>
<tr>
<td>4</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>5</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>6</td>
<td>250</td>
<td>200</td>
</tr>
<tr>
<td>7</td>
<td>250</td>
<td>200</td>
</tr>
<tr>
<td>8</td>
<td>250</td>
<td>200</td>
</tr>
<tr>
<td>9</td>
<td>200</td>
<td>200</td>
</tr>
</tbody>
</table>

4.4 Boundary Conditions

Since the temperature distribution is a function of both position and time, there are boundary conditions for $x$ and $r$ (position) as well as time, $t$. There is a symmetry plane at the center of the laser beam as well as at the inlet of the mesh. Initially the water in the tank is at room temperature. The top of the mesh, as well as the end of the mesh, are far enough away that they are always at room temperature. The boundaries of the wedge mesh are indicated below in Figure 4.4. The boundary conditions used in the CFD simulations are then summarized below in Table 4.2.

Figure 4.4: The boundaries of the wedge mesh for CFD of laser heating
Table 4.2: Boundary conditions for CFD of laser heating

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Temperature</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet</td>
<td>fixedValue 293 K</td>
<td>fixedValue 0 m/s</td>
</tr>
<tr>
<td>back</td>
<td>fixedValue 293 K</td>
<td>fixedValue 0 m/s</td>
</tr>
<tr>
<td>frontWedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>backWedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>axis</td>
<td>empty</td>
<td>empty</td>
</tr>
<tr>
<td>top</td>
<td>fixedValue 293 K</td>
<td>fixedValue 0 m/s</td>
</tr>
</tbody>
</table>

4.5 Parameters and Properties

4.5.1 Fluid Properties

All of the fluid parameters used in this simulation were calculated [50] and are listed below in Table 4.3.

Table 4.3: Fluid parameters for CFD of laser heating

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal diffusivity (m²/s), $k$</td>
<td>$1.43 \times 10^{-7}$</td>
</tr>
<tr>
<td>Density (kg/m³), $\rho$</td>
<td>998</td>
</tr>
<tr>
<td>Specific heat capacity (J/kg*K), $c_p$</td>
<td>4180</td>
</tr>
</tbody>
</table>

4.5.2 Laser Parameters

The Nd:YAG laser that was used had a total pulse length of 10 ns, with a rise time of 5 ns. The energy emitted by the laser was estimated to be approximately 30 mJ, with a pulse length of 10 ns the laser power was calculated to be 3 MW. The initial beam diameter of the laser is approximately 6 mm. A beam expander enlarges the beam diameter to approximately 18 mm, and the beam is then focused with a 200 mm lens. This process leads to a beam radius at the focal volume, often referred to as the waist size, of 3.765 μm and a Rayleigh length of 83.67 μm, both of which were calculated as described in section 2.2.3. The wavelength of the laser is dependent on the laser used. All Nd:YAG lasers have a wavelength of 1064 nm; however in this experiment a frequency doubler was used to change the wavelength of the laser to 532 nm. The focal length generally refers to the distance between the laser beam as it leaves the laser and the focal point of
the beam. In this simulation however, the focal length will refer to the distance between the laser at the point that it enters the tank of water and the focal point, or minimum beam radius, of the laser in the water. The focal length is a value that was arbitrarily chosen for this simulation, it was chosen as half the length of the mesh so that the focal volume of the laser would be situated at the center of the mesh.

As previously discussed in section 2.2.2, for this case the reflectivity and absorption coefficient are both dependent on the medium as well as the wavelength of the laser. The reflectivity is a unitless coefficient that has a value of 0.012 for a 530 nm laser in water [52]. The absorption coefficient of tap water is a much harder value to accurately determine; the literature indicates that for pure water at a wavelength of 532 nm the absorption coefficient ranges between 0.04 - 0.05 \(1/m\) [53]. However, the absorption coefficient for tap water is usually much higher and is closer to the absorption coefficient for open ocean than for pure distilled water [53]. The values for natural water also have a wide range of coefficients, a predicted value for a wavelength of 530 nm in natural water is \(a_w = 0.09784\) \(1/m\) [52]. A further complication is that the absorption coefficient of water is increased during laser-induced breakdown, when the heating of the water by the laser causes the water to vaporize leading to plasma formation and an increase in the absorption coefficient [53, 54]. A more complete discussion of the absorption coefficient can be found in section 4.5.3.

A summary of the laser parameters used are listed below in Table 4.4.

Table 4.4: Laser parameters for CFD of laser heating

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laser Power (MW), (E_p)</td>
<td>3</td>
</tr>
<tr>
<td>Beam Waist ((\mu m)), (w_0)</td>
<td>3.765</td>
</tr>
<tr>
<td>Reflectivity, (R_w)</td>
<td>0.012</td>
</tr>
<tr>
<td>Absorption coefficient (1/m), (a_w)</td>
<td>5</td>
</tr>
<tr>
<td>Laser rise time (ns), (a)</td>
<td>5</td>
</tr>
<tr>
<td>Rayleigh length ((\mu m)), (z_0)</td>
<td>83.7</td>
</tr>
<tr>
<td>Focal length (m), (L)</td>
<td>0.15</td>
</tr>
<tr>
<td>Wavelength of laser ((\mu m)), (\lambda)</td>
<td>532</td>
</tr>
</tbody>
</table>
4.5.3 The Absorption Coefficient

In order to obtain a more accurate estimate of the correct absorption coefficient, a simple experiment was conducted with a continuous wave Argon-ion laser. The Argon-ion laser emits at a range of wavelengths between 457.9 nm and 528.7 nm, has a beam diameter of 1.5 mm and a maximum output of 5 W. In comparison to the Nd:YAG laser, the minimum and maximum power output of the Argon-ion laser is much lower, and can therefore be tested using a power meter. The same tank of water used in the previous experimental setup was once again employed, this time however the Argon-ion laser was used without a beam expander or lens to focus the beam. Instead the beam shown through the tank at a constant beam radius, and the beam power was measured using the power meter at both the front and back of the tank. Assuming that a laser beam entering a tank of water at an incident angle of zero has a negligible angle of refraction, then neither the air nor the glass that the beam travels through before entering the water will have a significant affect on the absorption coefficient. Laser absorption tests with the same tank and air confirmed a negligible power attenuation within the uncertainty of the measurements. The absorption coefficient of the water can be determined from the difference in measured laser power over a certain distance.

The tank that will be used in the experiment is a rectangular tank with outer dimensions of 6" x 12" and inner dimensions of 5\(\frac{3}{4}\)" x 11\(\frac{3}{4}\)". The experiment will be conducted with both tap water and distilled water, as well as with distilled water with varying amounts of LDV (Laser-Doppler Velocimetry) seeding particles. The seeding particles will be used to determine the effect that the amount of particulates has on the absorption coefficient of water. It has been shown that increasing the laser energy can increase the absorption coefficient [53], therefore measurements will be taken at two different energy outputs. The percent of particulates in the water will be calculated using the volume of the water and the volume of the seeding particles. The seeding particles have a specific gravity of 1.1, and the volume of the particulates will be varied by varying the number of particles in the defined volume of water. The number of particles can be accurately determined by measuring the mass of the seeding particles.

The volume fraction of the particulates is given by equation (4.4).
\[ \phi = \frac{V_{pr}}{V_w} \]  \hspace{1cm} (4.4)

Where \( V_w \) is the volume of the water in the tank, 400 mL or 0.0004 m\(^3\), and \( V_{pr} \) is the volume of the total seeding particles in the water. Since the seeding particles are extremely small, the simplest method to vary the level of particulates is to measure the mass of the total amount of seed particles added to the water. The total volume of the particulates, \( V_{pr} \), can then be calculated using the total mass and the density of the particulates.

\[ V_{pr} = \frac{m_{pr}}{\rho_p} \]  \hspace{1cm} (4.5)

The Beers-Lambert law gives a relationship between the initial irradiance, \( I_0 \), and the irradiance at a certain position, \( I(x) \) [55].

\[ I = I_0 e^{a_w x} \]  \hspace{1cm} (4.6)

The absorption coefficients were then calculated for each test, as shown below in Table 4.5. The data shows, as expected, that an increase in the number of particulates in the water increases the absorption coefficient.

<table>
<thead>
<tr>
<th>Run</th>
<th>( a_w ) (1/m)</th>
<th>Run</th>
<th>( a_w ) (1/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low Power</td>
<td></td>
<td>High Power</td>
<td></td>
</tr>
<tr>
<td>Distilled Water</td>
<td>0.86</td>
<td>Distilled Water</td>
<td>0.92</td>
</tr>
<tr>
<td>( \phi = 0.000455 )</td>
<td>9.32</td>
<td>( \phi = 0.000455 )</td>
<td>8.93</td>
</tr>
<tr>
<td>( \phi = 0.000911 )</td>
<td>18.6</td>
<td>( \phi = 0.000911 )</td>
<td>18.6</td>
</tr>
<tr>
<td>( \phi = 0.001366 )</td>
<td>28.1</td>
<td>( \phi = 0.001366 )</td>
<td>28.5</td>
</tr>
<tr>
<td>( \phi = 0.001822 )</td>
<td>36.6</td>
<td>( \phi = 0.001822 )</td>
<td>37.0</td>
</tr>
</tbody>
</table>
Figure 4.5 illustrates that while the data did reveal a slightly higher absorption coefficient for higher laser powers, the difference between the absorption coefficient calculated for the lower power and the higher power was not significant enough to be definitive. Tap water was also measured at low power, however, there was not a significant difference between the measurements for the tap water and that of the distilled water. It could be possible that impurities were introduced into the distilled water during the experimental process, or it could be that double distilled water or deionized water would have given a closer measurement to the values of the absorption coefficient indicated in the literature. From the above analysis, the assumption can be made that the absorption coefficient for the tap water used in the experiment described in section 2.2.1 is on the order of 1 1/m.

4.6 Computational Parameters

Although the numerical solution to equation (2.1) is a transient solution, an adjustable time step was not used because the time step, by necessity, needed to be extremely small in order to accurately model the laser pulse. A fixed time step of $1 \times 10^{-11}$ s was chosen, a choice that will be validated in a time refinement study in section 4.7.3.
Table 4.6: Numerical schemes for CFD of laser heating

<table>
<thead>
<tr>
<th>Time Schemes</th>
<th>Euler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gradient Schemes</td>
<td>Gauss linear</td>
</tr>
<tr>
<td>Divergence Schemes</td>
<td>Gauss limitedLinear 1</td>
</tr>
<tr>
<td>Laplacian Schemes</td>
<td>Gauss linear corrected</td>
</tr>
<tr>
<td>Interpolation Schemes</td>
<td>Linear</td>
</tr>
<tr>
<td>Surface normal gradient schemes</td>
<td>Corrected</td>
</tr>
</tbody>
</table>

The numerical schemes that were used are summarized above in Table 4.6. An implicit first order Euler scheme was used for the time derivatives, while Gaussian schemes were used for the gradient, divergence and laplacian terms. There is a slight variation in the Gaussian schemes; Gauss linear uses central differencing, while Gauss limitedLinear uses limited linear differencing [47].

4.7 Analysis

4.7.1 Effect of Varying Laser and Fluid Parameters

While the absorption coefficient in tap water was experimentally estimated to be on the order of 1 1/m in section 4.5.3, the results of the experiment were not conclusive enough to determine a precise absorption coefficient. In the experiment described in section 2.2.1, it was determined that the Nd:YAG laser could produce a laser-generated cavitation bubble. In order for a cavitation bubble to form the laser must be heating the liquid to its boiling point, which is 373 K for water [50].

Since it is impossible to measure the energy output of the Nd:YAG using standard instrumentation, a precise measurement cannot be made. The maximum laser energy output of the laser used was 120 mJ, it was estimated that a cavitation bubble formed between 10 mJ and 30 mJ for this particular experimental setup. Therefore in order to estimate an appropriate laser energy output and absorption coefficient, several test cases were run varying laser energy and absorption coefficient. The temperature was measured at the center of the focal volume \((x = 0.15 \text{ m}, y = 0)\) and was plotted in Figure 4.6 versus the absorption coefficient for three different laser energies.
Since neither the absorption coefficient nor the laser energy are known precisely, they both must be estimated. Figure 4.6 shows that a laser energy of 30 mJ yields a temperature at the center of the focal volume above the boiling point of water. It is crucial that the temperature exceeds the boiling point of water in order for the bubble to form, therefore a laser energy of 30 mJ and an absorption coefficient of 5.0 1/m were used for the remaining analysis. The test runs showed that changing the laser input energy and the absorption coefficient did not significantly affect the temperature profile, just the magnitude of the temperature increase at the focal volume.

4.7.2 Grid Refinement Study

As previously discussed in section 4.3, the mesh that will be used is an axisymmetric “wedge” mesh, which has been refined in the center of the mesh where the focal volume of the laser will be. Since the beam radius at the focal volume is on the order of microns, the mesh must be very refined at the focal volume. A grid study was undertaken to ensure that the mesh was refined enough to capture the temperature gradients occurring in the focal volume. The temperature was evaluated for three different meshes; a coarse grid, a medium grid and a fine grid.
Table 4.7: Grid refinement study: comparison of the temperature at $x = 0.15$ m and $r = 0$ m

<table>
<thead>
<tr>
<th>Mesh Refinement</th>
<th>$T$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>392.773</td>
</tr>
<tr>
<td>Medium</td>
<td>392.937</td>
</tr>
<tr>
<td>Fine</td>
<td>393.006</td>
</tr>
</tbody>
</table>

As can be seen in Table 4.7, the temperature at a specific point does not change much as the grid is refined. A second order linear Gaussian scheme was used for both the laplacian and the divergence schemes. An order of convergence, $p$, was calculated using the following equation, where $\phi_h$ is the solution of the finest grid [45].

$$ p = \frac{\log \left( \frac{\phi_{2h} - \phi_{4h}}{\phi_h - \phi_{2h}} \right)}{\log 2} \quad (4.7) $$

A second order convergence was calculated for the temperature values listed above; the expected order of convergence for a second order scheme.

4.7.3 Time Refinement Study

An accurate solution in time was imperative because a transient solution was required. A time study will be performed in order to determine an appropriate time step using a first order implicit Euler scheme. Since the pulse of the laser is on the order of nanoseconds, the absolute maximum time step allowable would be a nanosecond. A time step of 0.01 ns was chosen to begin with, and a time study was conducted to validate this choice. The time step was varied by both doubling and halving the time step respectively. The temperature was then evaluated at the same specific location for each of the different time steps. The results are shown below in Table 4.8, and as expected there is some variation in the temperature as the time step is refined.
Table 4.8: Time study: comparison of the temperature at \( x = 0.15 \) m and \( r = 0 \) m

<table>
<thead>
<tr>
<th>Time Step ( \Delta t )</th>
<th>( T ) (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.02 ns</td>
<td>405.952</td>
</tr>
<tr>
<td>0.01 ns</td>
<td>346.741</td>
</tr>
<tr>
<td>0.005 ns</td>
<td>318.546</td>
</tr>
</tbody>
</table>

In a grid refinement study the order of convergence is calculated from the solution at a specific location for three different meshes of decreasing grid spacing. Similarly in a time refinement study the order of convergence is calculated from the solution at a specific location for three incremental time steps. The temperature values in Table 4.8 were used to calculate an order of convergence equal to one using equation (4.7). Since the order of convergence should match the order of the scheme and a first order time scheme was used, this indicates that \( \Delta t = 0.01 \) ns is an appropriate time step for these simulations. It should be noted that a second order scheme, such as Crank-Nicolson, could not be used for this problem because of the small time steps. The time steps were small by necessity, however, attempting to use a second order scheme resulted in an order of convergence of one instead of a convergence of two because of machine precision.

### 4.7.4 Temperature Profiles

The temperature profile that is expected is one in which the focal volume retains a somewhat ellipsoidal shape, while the temperature in the focal volume exceeds the boiling point. Since standard atmospheric conditions were assumed, the boiling temperature of water will be 100°C or 373 K. From the figures it can be seen that the temperature does indeed exceed the boiling temperature in the focal volume. Figure 4.8 shows the evolution of the temperature distribution at the center of the mesh, which is the center of the focal volume. Figure 4.8a is the temperature distribution along the focal length at a radius very close to zero, while Figure 4.8b is the temperature distribution along the focal radius at the center of beam at \( x = 0.15 \) m. The locations where the temperature was sampled are shown graphically in Figure 4.7.
Figure 4.7: Locations in the mesh where temperature is being sampled

(a) Temperature at $r = 0.1$ microns
(b) Temperature at $x = 0.15$ m

Figure 4.8: Temperature distribution of the laser focal volume at different times

While Figure 4.8 illustrates that the temperature of the water is indeed hot enough for the water to boil, Figure 4.9 shows that the mesh is refined enough to appropriately model the sharp temperature gradients inside the focal volume. Both figures show a Gaussian distribution for the temperature, as expected, which is increasing with time during the pulse duration. The peak temperature is at the center of the focal volume.

The temperature distribution remains constant after 5 ns for an extended period of time. Since the thermal diffusivity of water is so small, it will take several orders of magnitude of time longer than the pulse duration for the water to return to its original temperature. Figure 4.10 shows the temperature profile of the focal volume at 10 ns, where $w_0$ is the waist size, or minimum beam radius, and $z_0$ is the Rayleigh length, or the length of the focal volume.
Figure 4.9: Point distribution of the temperature profile in the laser focal volume at different times

Figure 4.10: Contour plot of the temperature in the laser focal volume at $t = 10$ ns

4.8 Discussion

The goal of this section was to use a thermal model to estimate the temperature distribution generated by a Nd:YAG laser in water. The incompressible energy equation was used with a source term that modeled the heat generated by the laser. The model predicted that the temperature of the water would exceed 373 K; leading to the boiling of the water. Now that it has been verified that this model will predict an appropriate temperature distribution, this model could be integrated into the compressible flow solver described in Chapter 2. Although beyond the scope of this research this source term
could be integrated into a compressible flow solver which solves the mass, momentum and energy equations. The flow solver would then be able to accurately model both the formation and collapse of a laser-generated cavitation bubble.
Chapter 5

CFD of Bubble Dynamics

After conducting a simple analysis on the dynamics of cavitation bubbles using the Rayleigh-Plesset equation, a more sophisticated analysis was performed. A computational analysis of a single cavitation bubble was accomplished using a two-phase, compressible flow solver; where the mass and momentum equations were solved to yield the pressure and velocity fields. The formation and collapse of a single bubble was studied for both bubble collapse in a free field and bubble collapse near a wall. The CFD solution of bubble collapse near a wall revealed that the compressible flow solver was able to capture both the shock wave emitted upon collapse and the high-speed microjet that moves from the center of the bubble at collapse towards the wall.

5.1 Approach

Using the compressible, two-phase flow solver described in section 2.3.2, a computational simulation will be performed; modeling the formation and collapse of a single cavitation bubble in a free field. A comparison will be made between this computational solution and the solution of the Rayleigh-Plesset equation for a bubble with the same initial conditions. The CFD solution will then be compared with experimental data of laser-generated bubble formation and collapse. After confirming that the compressible flow solver can accurately model complex bubble dynamics, it will be used to analyze single bubble formation and collapse against a solid surface.
5.2 Governing Equations

A more complete discussion of the compressible multiphase flow solver used in this analysis can be found in section 2.3.1; the flow solver is a homogeneous flow model that solves three governing equations and three algebraic equations. The conservation of mass yields both the volume fraction equation and the pressure equation, while the conservation of momentum yields an equation for the velocity [37]. The homogeneous mixture model requires only a single conservation of mass and momentum, as the model solves for only one pressure and one velocity field.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (5.1)
\]

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \text{div}(\rho \mathbf{u} \otimes \mathbf{u}) - \text{grad} \mu_{\text{eff}} \cdot \text{grad} \mu_{\text{eff}} \Delta \mathbf{u} = -\text{grad} p_{d} - (\text{grad} \rho) \mathbf{g} \cdot \mathbf{x} \quad (5.2)
\]

The density of the mixture, or the total density \( \rho \) is determined using the volume fractions of the mixture [37].

\[
\rho = \alpha_{1} \rho_{1} + \alpha_{2} \rho_{2} \quad (5.3)
\]

In addition to the two conservation equations and the equation for the total density, two equations of state are also necessary in order to calculate the density of the liquid, \( \rho_{2} \), and the density of the gas, \( \rho_{1} \).

\[
\rho_{1} = \left( \frac{p}{a_{c}} \right)^{\frac{1}{\gamma}} \quad (5.4)
\]

\[
\rho_{2} = \rho_{0} + \psi(p - p_{0}) \quad (5.5)
\]

In summary, there are three governing equations; a mass balance, a volume of fraction equation and a pressure equation, all described in greater detail in section 2.3.2. In addition there are three algebraic equations and in total there are six unknown fields: \( \rho_{1}, \rho_{2}, \alpha_{1}, \alpha_{2}, p \) and \( \mathbf{u} \). Three of the fields will be solved using the three governing equations, \( \alpha_{1}, p \) and \( \mathbf{u} \), where the governing equations will be manipulated to yield a pressure equation, a velocity equation and a volume fraction equation [37]. The remaining three fields, \( \alpha_{2}, \rho_{1} \) and \( \rho_{2} \) will be solved using equations (5.3), (5.4), and (5.5) [37].
5.3 Mesh Generation

5.3.1 Mesh Generation for Single Bubble Collapse in a Free Field

The mesh for the simulation of the formation and collapse of a single bubble in a free field was generated in BlockMesh, the OpenFOAM meshing tool. An axisymmetric “wedge” mesh was constructed with an angle $\theta$ of five degrees, where the mesh was a section of a sphere, as illustrated below in Figure 5.1.

Figure 5.1: Axisymmetric wedge mesh for CFD of single bubble collapse in a free field

Figure 5.2: Mesh distribution of 200 cell x 200 cell wedge mesh for single bubble collapse in a free field

The mesh seen in Figure 5.2 is 200 cells by 200 cells with the cells expanding in both the $x$ and $y$ directions. The mesh is 1 cell wide in the $z$ direction in order to satisfy
OpenFOAM’s requirements for a two dimensional axisymmetric mesh. The grading of the mesh near the location of the bubble expansion and collapse is very refined in order to capture the rapid changes in the pressure and velocity gradients, while the grid spacing farther away from the maximum bubble radius gradually becomes larger as it approaches the free field boundary condition.

5.3.2 Mesh Generation for Bubble Collapse Near a Wall

Single bubble collapse near a solid surface was also investigated. For the CFD solutions of bubble collapse against a wall an axisymmetric wedge mesh will be used. As in section 5.3.1 the mesh has an angle $\theta$ of five degrees, unlike the mesh used for the free field collapse however, this “wedge” is a section of a cylinder. CFD simulations will be performed for both the collapse of a bubble above a solid wall as well as for the collapse of a bubble under a solid wall. Different meshes will be used in each scenario. The standoff parameter, as previously discussed, is typically used in experimental research on laser-generated cavitation bubbles to quantify different regions near the plate where certain collapse behavior can be expected. The standoff parameter is defined as $\gamma = s/R_{\text{max}}$ where $R_{\text{max}}$ is the maximum radius of the bubble and $s$ is the distance between the center of the initial bubble and the wall. The intention initially was to create a single mesh that could be used for all of the under plate cases and a single mesh that could be used for all of the above plate cases. However, it soon became clear that this was impractical, as the number of cells in the meshes with larger standoff distances became so great that the runtime of the solution was unreasonably long. Therefore a series of meshes were systematically created with the intention of producing very similar meshes; with the key difference being the length from the center of the bubble to the plate. The region where the initial bubble is located has the same number of grid points for each mesh, however the location of the initial bubble is different in each mesh. Similarly the same expansion ratios were used for each mesh, the key difference was the bubble location.

The distance between the bubble and the wall was varied for both bubble collapse under a plate and bubble collapse above a plate. The standoff parameters were chosen
with the intent to model the behaviors observed by Philipp and Lauterborn [17], who argued that a jet only causes damage to the plate at $\gamma \leq 0.7$ and that the first collapse only affects the wall at $\gamma < 1$ while the wall is affected by the second bubble collapse at $\gamma < 2$. After exposing a specimen of aluminum to 100 identical laser-generated bubbles, Philipp and Lauterborn determined that the greatest damage occurred at $\gamma \leq 0.3$ and at $1.2 < \gamma < 1.4$ [17]. As the standoff parameter $\gamma$ is varied, the only variable that changes is $s$, the distance between the center of the bubble and the wall. In order to handle the large pressure and velocity gradients as the bubble collapses, the mesh needs to be refined at the center of the bubble and near the wall. Therefore, a small constant mesh spacing was used in the initial diameter of the bubble, $\delta_{2R_0}$, and a slightly larger constant mesh spacing was then applied to the area between the wall and the bubble, $\delta_{(s-R_0)}$. The grid spacing was then expanded using a grid expansion ratio of 1.05 for the region between the initial bubble diameter and the estimated maximum bubble radius $R_{max}$. A grid expansion ratio of 1.1 was then used for the region between the estimated maximum bubble radius and the far field radius $R_\infty$.

![Figure 5.3: Mesh generation for single bubble collapse against a wall; not drawn to scale](image)

Figures 5.1 and 5.3 show the grid distribution of each grid that was used. Since the standoff parameter affects the bubble dynamics of the collapse, some exceptions had to be made for certain cases. For the three cases closest to the wall, $\gamma = 0.2$, 0.5 and 0.7,
a larger grid spacing between regions $s - R_0$ and $2R_0$ was required ($\delta_{(s-R_0)} \approx 1.15 \delta_{R_0}$) while for the two cases farthest away from the wall, $\gamma = 1.0$ and $1.3$, a slightly smaller grid spacing between the two regions was necessary ($\delta_{(s-R_0)} \approx 1.05 \delta_{R_0}$). The smaller grid spacing worked well for all of the cases, however for the cases where the bubble collapsed closer to the wall the small grid spacing resulted in an extremely long runtime. Additionally it was found that for the cases where the bubble is deforming significantly, the estimated $R_{\text{max}}$ should be larger than the predicted $R_{\text{max}}$, as the actual maximum radius is greater than the predicted $R_{\text{max}}$.

Table 5.1: Mesh distribution for different standoff distances $\gamma$

<table>
<thead>
<tr>
<th>$\gamma$</th>
<th>$2R_0$(m)</th>
<th>$R_{\text{max}}$(m)</th>
<th>$R_\infty$(m)</th>
<th>$\delta_{(s-R_0)}$(m)</th>
<th>Exp. Ratio #1</th>
<th>Exp. Ratio #2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>0.0002</td>
<td>0.002</td>
<td>0.05</td>
<td>7.69x10^{-6}</td>
<td>6.9x10^{-6}</td>
<td>1.05</td>
</tr>
<tr>
<td>0.5</td>
<td>0.0002</td>
<td>0.002</td>
<td>0.05</td>
<td>7.84x10^{-6}</td>
<td>6.9x10^{-6}</td>
<td>1.05</td>
</tr>
<tr>
<td>0.7</td>
<td>0.0002</td>
<td>0.002</td>
<td>0.05</td>
<td>7.89x10^{-6}</td>
<td>6.9x10^{-6}</td>
<td>1.05</td>
</tr>
<tr>
<td>1.0</td>
<td>0.0002</td>
<td>0.001</td>
<td>0.05</td>
<td>7.2x10^{-6}</td>
<td>6.9x10^{-6}</td>
<td>1.05</td>
</tr>
<tr>
<td>1.3</td>
<td>0.0002</td>
<td>0.001</td>
<td>0.05</td>
<td>7.2x10^{-6}</td>
<td>6.9x10^{-6}</td>
<td>1.05</td>
</tr>
</tbody>
</table>

5.4 Boundary Conditions

5.4.1 Boundary Conditions for Single Bubble Collapse in a Free Field

Recall that OpenFOAM computes every solution in three dimensions by default. In order to model an axisymmetric flow, two “wedge” boundary conditions must be used in order for OpenFOAM to solve the flow in two dimensions. The boundary conditions used in the CFD simulation of the formation and collapse of a single bubble in a free field are listed below in Table 5.3.

Table 5.2: Boundary conditions for CFD of single bubble formation and collapse in a free field

<table>
<thead>
<tr>
<th>Boundary</th>
<th>$\alpha_1$</th>
<th>$p_d$</th>
<th>U</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symmetric</td>
<td>symmetryPlane</td>
<td>symmetryPlane</td>
<td>symmetryPlane</td>
</tr>
<tr>
<td>Farfield</td>
<td>zeroGradient</td>
<td>zeroGradient</td>
<td>fixedValue (0 0 0)</td>
</tr>
<tr>
<td>WedgeA</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>WedgeB</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>defaultFaces</td>
<td>zeroGradient</td>
<td>zeroGradient</td>
<td>zeroGradient</td>
</tr>
</tbody>
</table>
The boundaries that these conditions are then imposed on are labeled in Figure 5.4.

Figure 5.4: Boundary patches for CFD of single bubble collapse in a free field

5.4.2 Boundary Conditions for Single Bubble Collapse Near a Wall

As with the boundary conditions for the collapse of a single bubble in a free field, all of the meshes created for the simulations of bubble collapse against a wall required two “wedge” type boundary conditions. The boundary conditions for $\alpha_1$, $p_d$ and $U$ are listed for each boundary in Table 5.3. The corresponding boundaries are labeled in Figure 5.5 for bubble collapse under the plate, and in Figure 5.6 for bubble collapse above the plate.

Table 5.3: Boundary conditions for CFD of single bubble collapse against a wall

<table>
<thead>
<tr>
<th>Boundary</th>
<th>$\alpha_1$</th>
<th>$p_d$</th>
<th>$U$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall</td>
<td>zeroGradient</td>
<td>zeroGradient</td>
<td>fixedValue uniform (0 0 0)</td>
</tr>
<tr>
<td>Farfield</td>
<td>inletOutlet uniform 0</td>
<td>zeroGradient</td>
<td>fixedValue uniform (0 0 0)</td>
</tr>
<tr>
<td>WedgeA</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>WedgeB</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>internalField</td>
<td>uniform 0</td>
<td>uniform 101325</td>
<td>uniform (0 0 0)</td>
</tr>
</tbody>
</table>
5.5 Fluid Properties

The fluid parameters used in the bubble collapse simulations are listed below in Table 5.4 and Table 5.5, where the gas parameters are for air [56] and the liquid parameters are for water [50]. These fluid properties will be used in both the CFD solutions of single bubble collapse in a free field, as well as in the simulations of bubble collapse against a solid wall.
Table 5.4: Gas parameters for CFD of single bubble formation and collapse

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kinematic viscosity (m²/s), ν</td>
<td>1.479 x 10⁻⁵</td>
</tr>
<tr>
<td>Specific heat capacity (J/kg*K), Cᵥ</td>
<td>720</td>
</tr>
<tr>
<td>Gas constant for air (J/kg*K), Rgas</td>
<td>287</td>
</tr>
<tr>
<td>Ratio of specific heats, κ</td>
<td>1.4</td>
</tr>
<tr>
<td>Density (kg/m³), ρ</td>
<td>1.2</td>
</tr>
<tr>
<td>Adiabatic Constant (kg⁻κ·m³κ·s⁻²), aₑ</td>
<td>101325</td>
</tr>
</tbody>
</table>

Table 5.5: Liquid parameters for CFD of single bubble formation and collapse

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kinematic viscosity (m²/s), ν</td>
<td>1 x 10⁻⁶</td>
</tr>
<tr>
<td>Initial density (kg/m³), ρ₀</td>
<td>1000</td>
</tr>
<tr>
<td>Initial pressure (Pa), p₀</td>
<td>101325</td>
</tr>
<tr>
<td>Speed of sound (m/s), c</td>
<td>1484</td>
</tr>
<tr>
<td>Temperature (K), T</td>
<td>288</td>
</tr>
<tr>
<td>Surface Tension (N/m), S</td>
<td>0.07</td>
</tr>
</tbody>
</table>

### 5.6 Computational Parameters

The computational parameters were chosen specifically for this problem. Since the solution was a transient solution it was imperative to choose a time step that would be small enough to yield an accurate temporal discretization. An adjustable time step was used in OpenFOAM, where the time step was adjusted based on the Courant number. The Courant condition for a flow that is slightly compressible, such as the formation and collapse of a cavitation bubble, is given as follows [45].

\[
\frac{c \Delta t}{\Delta x} < C₀
\]  

(5.6)

This yields a smaller time step than the Courant condition for an incompressible flow [45].

\[
\frac{u \Delta t}{\Delta x} < C₀
\]  

(5.7)

Simulations were tried with both Courant conditions, and although the acoustic Courant condition is the appropriate condition to use in this situation, the runtime of the solution was extremely slow when compared to the runtime of the solution using equation (5.7). Therefore, in order to reduce the runtime and computational expensive
of the simulation, equation (5.7) was used to determine the time step.

The numerical schemes used are summarized below in Table 5.6. An implicit first order Euler scheme was used for the time derivatives, while Gaussian schemes were used for the gradient, divergence and laplacian terms. There were some slight differences in the different divergence schemes; most however were either Gauss limited, a blended version of an unbounded second order scheme and a bounded first order scheme, or Gauss upwind, a first order unbounded scheme [47]. The same numerical schemes were used for all of the single bubble collapse simulations, including the simulations of bubble collapse against a wall.

Table 5.6: Numerical schemes for CFD of single bubble formation and collapse

<table>
<thead>
<tr>
<th>Time Schemes</th>
<th>Euler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gradient Schemes</td>
<td>Gauss linear</td>
</tr>
<tr>
<td>Divergence Schemes</td>
<td>Gauss</td>
</tr>
<tr>
<td>Laplacian Schemes</td>
<td>Gauss linear corrected</td>
</tr>
<tr>
<td>Interpolation Schemes</td>
<td>Linear</td>
</tr>
<tr>
<td>Surface normal gradient schemes</td>
<td>Corrected</td>
</tr>
</tbody>
</table>

5.7 CFD of a Single Cavitation Bubble in a Free Field

The formation and collapse of a single bubble was modeled in OpenFOAM using the computational method described in section 2.3.2. As previously discussed in section 3.5.3, laser-generated cavitation bubbles typically have a maximum radius on the order of 0.1 to 1 mm. There is still much debate in the literature about the minimum radius of a laser-generated cavitation bubble, where most of these inconsistencies can be attributed to the limitations of the optical and camera systems. For the purposes of this analysis it was assumed that the cavitation bubble had a minimum radius of 0.1 mm and a maximum radius of 1 mm. While this compression ratio is smaller than those seen by many experimentalists studying laser-generated cavitation bubbles [22, 26], a larger compression ratio is impractical from a computational perspective. The pressure and velocity differences may be smaller for a smaller compression ratio, as discussed in section 3.5.3, but the overall physics of the problem will be maintained without the extreme
computational cost required of a larger compression ratio. For a compression ratio of ten the initial conditions are provided below in Table 5.7, where the initial pressure of the gas inside the bubble was determined using the Rayleigh-Plesset equation.

Table 5.7: Initial conditions of single bubble collapse in a free field determined using the Rayleigh-Plesset equation

<table>
<thead>
<tr>
<th>$R_0$ (mm)</th>
<th>$R_{max}$ (mm)</th>
<th>$pG_0$ (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>1.0</td>
<td>$4.3 \times 10^7$</td>
</tr>
</tbody>
</table>

Of interest is the growth of the bubble, and the velocity and pressure profiles both inside the bubble and in the surrounding liquid. The radius of the bubble is estimated in the compressible flow model using the volume of gas in the two-phase flow. Shown below in Figure 5.7 is the profile of the radius as a function of time for the growth and collapse of a single bubble. As expected, the bubble initially expands to its maximum radius, collapses and oscillates back and forth until the water is once again pure liquid.

Figure 5.7: Radius versus time for the formation and collapse of a single free cavitation bubble

Figure 5.8 shows the nondimensional radius versus the nondimensional time of the bubble growth and collapse. As expected the nondimensional radius reaches a peak at ten, when it is equal to the compression ratio. Similarly, the first bubble collapse occurs at a nondimensional time of one, when the nondimensional time is equal to the collapse time. Slightly more than thirteen oscillations are reached before a nondimensional time of ten.
occurs; which is more than the Rayleigh-Plesset equation would have predicted. The Rayleigh-Plesset equation does not however, take into account the compressibility of the liquid, nor does it have any mechanism to dampen the size of the bubble oscillations.

![Figure 5.8: Nondimensional radius versus nondimensional time for the formation and collapse of a single cavitation bubble in a free field](image)

As mentioned previously, the radius used in the above graphs is an approximate bubble radius calculated using the volume fraction of the gas. Initially the bubble is spherical, so this approximation is accurate. However, the bubble eventually loses its symmetry and the volume fraction of the gas becomes a much better indicator of the bubble shape than the calculated radius. As shown below in Figure 5.9, the volume fraction of the gas phase is a good visual indicator of the shape of the bubble. Figure 5.9b shows the bubble at its maximum radius, when the bubble is still spherical and its core is comprised entirely of gas. Figure 5.9c shows the bubble at its minimum radius, at approximately the first collapse, while Figure 5.9d shows that the bubble is increasing during the second oscillation while still maintaining a spherical shape. Figure 5.10 shows the pressure of the bubble and the surrounding liquid at different times during the formation and collapse of the bubble. Figures 5.10a and 5.10c show the pressure of the bubble at the initial instant of formation and at the first bubble collapse. The high initial pressure of the gas inside of the bubble leads to the growth of the bubble, similarly the high pressure at $t = 190 \mu s$ leads to the growth of the rebound bubble. Figure 5.10b
and 5.10d show the pressure of the bubble when the first and second bubble oscillations are at their maximum radius. The pressure inside of the bubble is very low, while the pressure of the surrounding liquid is much higher, this pressure differential leads to the collapse of the bubble in both instances.

Figure 5.9: Volume fraction of the gas phase, $\alpha_1$, at different times during the first formation and collapse of a single bubble in a free field
5.7.1 Comparison with solution of the Rayleigh-Plesset equation

The analysis to this point has focused on using a compressible finite volume method to computationally model the formation and collapse of a single bubble. While this model is more complex than the Rayleigh-Plesset equation, it is expected that there should be some similarities between the solutions of the two models. As seen in Figure 5.11, the CFD solution and the solution of the Rayleigh-Plesset equation result in an almost identical shape for the first bubble oscillation. Since the assumptions of the Rayleigh-Plesset equation are only valid for the first bubble oscillation, the solution was not compared to the CFD results after the first collapse.
It is interesting that the solutions are similar during the first collapse, as compressibility effects begin to occur after the maximum radius is reached. This could be due to the smaller compression ratio. A larger compression ratio may yield a similar shape during the formation of the bubble, but the collapse times of the two models should be slightly different as the Rayleigh-Plesset equation typically overpredicts collapse time. For this case however, there is remarkable similarity between the collapse time of the two models, as shown below in Table 5.8.

<table>
<thead>
<tr>
<th></th>
<th>$R_0$ (mm)</th>
<th>$p_{G_0}$ (Pa)</th>
<th>$R_{max}$ (mm)</th>
<th>$\tau_C$ (\textmu s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>0.1</td>
<td>$4.3 \times 10^7$</td>
<td>1.003</td>
<td>188.9</td>
</tr>
<tr>
<td>Rayleigh-Plesset equation</td>
<td>0.1</td>
<td>$4.3 \times 10^7$</td>
<td>1.006</td>
<td>190.8</td>
</tr>
</tbody>
</table>

As expected, for the same initial radius and initial gas pressure inside the bubble, the maximum size of the bubble is very close in both the Rayleigh-Plesset equation and the CFD solution. As previously discussed, taking into account the relatively small compression ratio, the collapse times of the two models are very similar. A comparison between the two models verified that the compressible flow model does behave as expected, it also illustrated how well, despite its many approximations, the Rayleigh-Plesset equation
estimates the shape of a cavitation bubble during the initial formation and collapse.

### 5.7.2 Comparison with Experimental Data

After confirming that there is reasonable agreement between the CFD results and the solution of the Rayleigh-Plesset equation, a comparison will now be made between the CFD simulations and experimental data taken from Müller et al. [26]. Müller et al. generated cavitation bubbles with a maximum radius of $R_{\text{max}} = 746.9 \, \mu m$ using a Nd:YAG laser [26]. For the purpose of comparing the CFD model and the experimental data, three different initial pressure conditions will be analyzed. Müller et al. estimated that at the maximum bubble radius the pressure inside of the gas will be $P_{G_0} = 4.579 \, \text{Pa}$ [26]. Zein compared the results of a seven-equation multiphase model with the experimental data of Müller et al. assuming that the gas pressure at the maximum radius is $P_{G_0} = P_v$, or approximately $2330 \, \text{Pa}$ [31]. A CFD solution will be found using these two different initial conditions, where the bubble is initially at its maximum radius. An additional CFD simulation will be carried out starting at the formation of the bubble, where the initial conditions are based on the solution of the Rayleigh-Plesset equation, for a bubble with a maximum radius of $R_{\text{max}} = 746.9 \, \mu m$. All three solutions will then be compared to the experimental data.

![Comparison of CFD solution and experimental data](image)

**Figure 5.12:** Comparison of CFD solution and experimental data where the initial conditions are $P_{G_0} = 5.21 \times 10^7 \, \text{Pa}$ and $R_0 = R_{\text{min}} = 6.29 \times 10^{-5} \, \text{m}$
Little is known about the pressure inside of a cavitation bubble because it is extremely difficult to measure. CFD simulations of bubble collapse therefore require an approximation of the initial pressure conditions. Figure 5.12 is a comparison between the calculated radius for the CFD solution of bubble collapse, using initial conditions determined from the solution of the Rayleigh-Plesset equation, and experimental data. There is a very good correlation between the CFD solution and the experimental data during the initial formation and collapse of the bubble, especially between the maximum radius and the collapse time. However, the size of the rebound bubble is noticeably overpredicted by the CFD solution.

Assuming that the pressure inside of the bubble is equal to the vapor pressure at the maximum radius of the bubble, the collapse of the bubble was modeled starting at $R_{\text{max}}$. Figure 5.13 shows the evolution of the bubble radius as a function of time for both the CFD solution and the experimental data. The CFD solution for the collapse of the first bubble compares well to the experimental data, however, once again the size of the rebound bubble is overestimated by the CFD solution.

Using the initial conditions calculated by Müller et al. [26], Figure 5.14 shows that an initial pressure below the vapor pressure does indeed result in a smaller rebound bubble. However, the radius of the rebound bubble is not as spherical as the radius predicted by
either the experimental data or the previous two CFD simulations. Although changing
the initial conditions of the CFD solution can reduce the size of the rebound bubble,
it appears that the initial conditions alone cannot account for the overprediction of the
radius of the rebound bubble.

Figure 5.14: Comparison of CFD solution and experimental data where the initial
conditions are $P_{G0} = 4.579$ Pa and $R_0 = R_{max} = 746.9 \mu$m

In a laser-generated bubble the bubble is created when the energy of the laser causes
the water to vaporization at the focal volume of the laser; this rapid fluctuation in tem-
perature will result in a temperature gradient across the liquid-vapor interface, something
not accounted for in the CFD model used in these simulations. Adding a phase change
model that could account for the evaporation and condensation across the interface of
the two phases, as well as model the heat transfer resulting from the temperature rise,
could result in a CFD model that would more accurately model the rebound bubble.
Zein [31] and Dreyer et al. [28] both compared various models with the experimental
data of Müller et al. [26] and found that the best correlation between the results of the
their models and the experimental data occurred when they accounted for phase change.
They also found that when the effects of mass transfer were accounted for, a rebound
bubble only formed when they assumed that there was inert gas inside of the bubble
[28, 31].
5.7.3 Grid Refinement Study

A grid refinement study was conducted to ensure that the mesh was refined in an appropriate manner. Three different meshes were used; a coarse mesh, a medium mesh and a fine mesh. The solution was evaluated at three different times, \( t_1 = 60 \, \mu s \), \( t_2 = 90 \, \mu s \) and \( t_3 = 120 \, \mu s \), as indicated in Figure 5.15. The volume fraction of the gas, \( \alpha_1 \), and the pressure, \( p \), were both evaluated as a function of the position \( r \) for each of the three meshes, at the three different times. As can be seen in Figure 5.15 and Figure 5.16, the maximum bubble radius decreased with a finer grid, a behavior also observed both by Miller et al. [37] and Zein [31] as they decreased the grid spacing in CFD simulations of bubble collapse. Zein inferred in his thesis that the radius was decreasing with an increase in the number of cells because of the diffusion at the interface [31].

![Figure 5.15: Bubble radius versus time for three different meshes with t1, t2 and t3 indicated](image)

An analysis of the grid study was complicated by the complexity of the solution. Since the CFD analysis was conducted using a compressible multiphase flow solver that resulted in a transient solution, it was difficult to choose a point in time and space where the bubble was experiencing the same behavior in each mesh. Figure 5.15 shows that at time \( t_1 \), for the finest grid the bubble is approximately at its maximum radius, while for the medium grid and the coarsest grid the bubble is still expanding. Therefore, at \( t_1 \) the pressure and velocity will be significantly different in all three meshes because the
bubble is at a different part of the formation and collapse process in all three meshes. In addition, while in theory the grid can be very refined, in practice there is a threshold where the grid spacing becomes so small that with a constant Courant condition the time step of the solution becomes tiny and the solution will take an unreasonable amount of time to complete. The only practical recourse is to graphically examine the solution for all three grids. Figure 5.16 shows the volume fraction of the gas versus the radial coordinate \( r \), where each figure shows \( \alpha_1 \) from the center of the bubble through the vapor-liquid interface and into the surrounding liquid. It appears that the slope of the line as the void fraction transitions from the gas phase to the liquid phase is similar for all three meshes, and the magnitude of the difference in void fraction between the coarse mesh and medium mesh is approximately the same as the difference between the medium mesh and the fine mesh. The difference in \( \alpha_1 \) between the three meshes does seem to increase after the bubble collapse, which agrees with the results of Zein [31] who also found an increased difference in the volume fraction of the gas after the collapse.

Figure 5.17 shows the pressure as a function of the radial direction \( r \), at the same three times shown in Figure 5.16. This figure shows the pressure over a greater distance \( r \) than the previous figure, extending from the center of the bubble at \( r = 0 \) mm through the bubble interface and well into the far field region at \( r = 150 \) mm. In contrast to the discrepancies shown in Figure 5.16, Figure 5.17 shows that the difference in pressure is relatively small with two exceptions; as the pressure is approaching atmospheric pressure and farther into the far field, when the pressure wave travels outwards from the bubble into the surrounding fluid. The pressure is slightly higher for the finest mesh up until the pressure wave, when the pressure becomes greater for the coarsest mesh.
Figure 5.16: Volume fraction of the gas versus the radial coordinate $r$ at three different times for three different meshes
Figure 5.17: Pressure versus the radial coordinate $r$ at three different times for three different meshes
5.8 Single Bubble Collapse Near a Wall

After analyzing the collapse of a single cavitation bubble in a free field, the collapse of a single cavitation bubble against a solid wall will be investigated using the same initial conditions as the free bubble case. Different standoff distances were examined and the trends observed were compared to the trends observed in the literature on laser-generated cavitation bubbles. In addition to the effect of standoff distance on bubble collapse, the effect of gravity is also of interest, therefore a comparison was made between the collapse of a bubble above a wall and the collapse of a bubble under a wall.

5.8.1 Effect of Standoff Distance on Bubble Collapse against a Wall

Without accounting for the effects of gravity or surface tension, the effect of standoff distance on bubble collapse was examined. The distance between the bubble at its initial position and the wall is of interest because previous research [17, 18, 19, 21, 22, 33, 36] has indicated that it has significant influence on the behavior of the bubble collapse. Five different standoff distances were chosen, varying from $\gamma = 0.2$ to $\gamma = 1.3$, with the only change in each of the simulations being the distance to the wall. All of the simulations described in this section were conducted for the collapse of a bubble above a plate.

Figure 5.18 shows the pressure as a function of time at two locations in the mesh, the first is at the center of the initial bubble, $y = s$, and the second is at a location along the wall. The first pressure pulse shown does not necessarily model the pressure pulse that would be seen in a laser-induced cavitation bubble. In a laser-induced cavitation bubble the bubble forms because the liquid is being vaporized, leading to a rapid temperature increase. This rapid temperature increase would also result in an increase of pressure, however the first pressure pulse that appears in Figure 5.18 is not the pressure pulse that corresponds to the formation of a laser-induced cavitation bubble. The initial pressure condition was determined from the solution of the Rayleigh-Plesset equation. Since a typical laser-generated cavitation bubble has a maximum bubble radius of approximately 1 mm, the initial pressure of the gas used in the CFD simulations was the same as the initial pressure necessary for the bubble radius to grow to 1 mm in the solution of the
Rayleigh-Plesset equation.

The dynamics of bubble collapse are similar for each standoff distance, the initial high pressure inside of the bubble causes it to expand until the bubble reaches its maximum size. The bubble then rapidly collapses, upon collapse a high pressure pulse is emitted and a microjet travels from the center of the bubble to the wall. The bubble then expands a second time, this time with a much smaller maximum radius, and collapses a second time, leading to yet another high pressure pulse. The second pressure pulse shown in Figure 5.18 corresponds to the first collapse of the bubble, while the third pressure pulse corresponds to the second bubble collapse. Figure 5.18 shows that both pulses are much higher for $\gamma = 0.2$ than for the other standoff distances. This could be because the bubble collapse is so close to the wall. Phillip et al. found that the most damage was produced to a solid surface for $\gamma \leq 0.3$ and for $\gamma$ between 1.2 and 1.4 [17]. The high pressure peaks at collapse for the bubble at $\gamma = 0.2$ seem to confirm this conclusion. Figure 5.18 also illustrates that the collapse time is slightly different for each standoff distance. This is a result of the wall influencing the dynamics of the collapse, deforming the bubble and delaying collapse for cases close to the wall.

![Figure 5.18: Comparison of the pressure at different probe locations for different standoff distances](image)

Comparing the pressure fields for all five standoff distances is useful because it does confirm the assumption that standoff distance will affect the dynamics of the bubble collapse. In order to more closely examine the bubble dynamics of each standoff distance,
the volume fraction of the gas and the pressure field will be evaluated at each standoff distance. For the case where the standoff distance is the smallest, $\gamma = 0.2$, the evolution of the volume fraction in time is presented below in Figure 5.19. The bubble reaches its maximum size at approximately 100 $\mu$s, and collapses at approximately 210 $\mu$s. The bubble then expands a second time and achieves a maximum radius at 320 $\mu$s; rapidly collapsing a second time.

Figure 5.19: Volume fraction of the gas for $\gamma = 0.2$ at different times in the bubble formation and collapse
Figure 5.20 is a series of contour plots at time steps of interest. At 10 µs, Figure 5.20a, it is clear that a pressure wave is propagating outwards from the center of the bubble as the bubble expands. Figure 5.20b shows that when the bubble is at its maximum size the pressure inside of the bubble is much lower than atmospheric pressure, a driving factor in the collapse of the bubble. Figure 5.20c is taken just after the first collapse of the bubble, the pressure is once again very high in the center of the bubble enabling the bubble to expand to a second maximum radius as seen in Figure 5.20d. Figure 5.20e shows that the pressure in the bubble is once again very high after the second bubble collapse.

For a standoff distance of $\gamma = 0.2$ the bubble is collapsing very close to the wall, which means that the high pressure values seen in Figure 5.20c and Figure 5.20e correspond to high impact pressures at the wall. Figure 5.21a shows the pressure at the wall, $y = 0$, at the same times shown in Figure 5.20c and Figure 5.20e; the first and second bubble
collapses respectively. Figure 5.21b shows the pressure at $y = 0$ at different $x$ locations as a function of time. It is apparent that the first bubble collapse results in a very large impact pressure on the wall near the center of the bubble radius, at 0.1 mm the pressure is approximately 150 times larger than atmospheric pressure. Figure 5.21 shows that the second collapse results in a much lower impact pressure than the first collapse, although this pressure pulse is once again significantly greater than atmospheric pressure.

Figure 5.21: Change in pressure for CFD of bubble collapse at $\gamma = 0.2$ along the wall at different times and $x$ locations

Bubble collapse is characterized not only by the intense pressure pulses emitted upon collapse but also by the microjet that is formed during the initial bubble collapse. Figure 5.22a shows the magnitude of velocity at the initial bubble center as a function of time. The velocity at the location of the initial bubble center increases rapidly as the bubble collapses, just after 200 $\mu$s.
(a) Magnitude of velocity at the center of the bubble $x = 0$ and $y = s$

(b) Magnitude of velocity at $x = 0$ along $y$ at different times

Figure 5.22: Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 0.2$ at the center of the bubble and as it travels towards the wall

Figure 5.22b shows the magnitude of the microjet formed at the center of the bubble; which upon collapse migrates outward towards the wall, where $y = 0$ is the location of the wall. The magnitude of the velocity decreases rapidly as it approaches the wall because there is a no slip boundary condition being imposed upon the wall. The bubble dynamics characteristic of laser-generated bubble collapse have all been illustrated in Figure 5.19, Figure 5.20, Figure 5.21 and Figure 5.22; including the first and second collapse of the bubble as well as the high pressure pulse and microjet emitted upon the collapse of the bubble. The CFD results for a standoff distance of $\gamma = 0.5$ are similar, although there are slight disparities worth mentioning.
Figure 5.23: Volume fraction of the gas for $\gamma = 0.5$ at different times in the bubble formation and collapse

Figure 5.23 shows the time history of the volume fraction of the gas as the bubble forms and collapses for a standoff distance of $\gamma = 0.5$. At the bubble’s maximum radius, 100 $\mu$s, it is clear that the bubble is farther away from the wall than in Figure 5.19. In addition, the first bubble collapse occurs sooner than it did for $\gamma = 0.2$, closer to 210 $\mu$s. The expansion of the bubble after the first collapse is also much different for this case, Figure 5.23g shows that the bubble does not expand spherically as it did in Figure 5.19h.
but rather has become toroidal. Philipp and Lauterborn experimentally observed that
the bubble becomes toroidal after collapse because of the influence of the microjet as
it travels through the center of the bubble towards the wall [17]. It is probable that
this is actually the expected behavior of the bubble after collapse, and that the different
dynamics observed when $\gamma = 0.2$ were due to the bubble’s close proximity to the wall.

Figure 5.24 is a series of contour plots of the pressure as the bubble forms and
collapses. Once again a pressure wave is observed at 10 $\mu$s as the vapor bubble expands
in the liquid, and the pressure inside of the bubble is the lowest at the bubble’s maximum
radius at 100 $\mu$s. Figures 5.24c and 5.24d indicate that the bubble loses much of its
symmetry upon collapse, and Figure 5.24e confirms that the bubble does indeed become
toroidal after the first collapse. In comparison to Figure 5.21, Figure 5.25 shows that the
magnitude of the pressure pulses emitted at the first and second collapse of the bubble
are much smaller for $\gamma = 0.5$ than for $\gamma = 0.2$. The magnitude of the jet velocity however,
is much higher for $\gamma = 0.5$ than for $\gamma = 0.2$ as can be seen in Figure 5.26.

Figure 5.25: Change in pressure for CFD of bubble collapse at $\gamma = 0.5$ along the wall at different times and $x$ locations

Figure 5.26: Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 0.5$ at the center of the bubble and as it travels towards the wall

Similarly, for bubble collapse when $\gamma = 0.7$, the pressure wave emitted upon the collapse of the bubble is not as strong as for the case where $\gamma = 0.2$. However, the magnitude of the jet velocity was higher, in the same range as for the case where $\gamma = 0.5$. Figure 5.27 is a series of contour plots of the volume fraction of the gas as the bubble forms and collapses. The bubble dynamics observed are very similar to the those observed in Figure 5.23, where $\gamma = 0.5$. Figure 5.28, contour plots of the pressure, show behavior comparable to the pressure plots for $\gamma = 0.5$. 

Figure 5.27: Volume fraction of the gas for $\gamma = 0.7$ at different times in the bubble formation and collapse.
Figure 5.29a shows the pressure along the wall at both the initial time and the two collapse times. While Figure 5.29b shows the change in the pressure as a function of time at different $x$ locations along the wall. Figure 5.29b shows that the pressure pulse at the second collapse is greater than the pressure pulse at the first collapse at certain $x$ locations; the reverse of what was seen when $\gamma = 0.2$ and $\gamma = 0.5$. This could be because the initial collapse is occurring farther away from the wall while the second collapse is occurring closer to the wall.

Figure 5.28: Contour plot of pressure for $\gamma = 0.7$ at different times in the bubble formation and collapse; note that pressure has been rescaled at each time step.
For a standoff distance of $\gamma = 1$, the bubble should expand just to the wall, collapse, and then move towards the wall as the microjet quickly moves from the center of the bubble towards the wall. Figure 5.31 illustrates that this does indeed happen. Figure 5.31c shows that the maximum bubble radius occurs at 100 $\mu$s, as it did for the previous cases, but has a slightly shorter collapse time, as can be seen in Figure 5.31d and 5.31e. Another interesting phenomenon revealed in Figure 5.31 is what is referred to as the “splash” effect. The “splash” effect occurs for standoff distances in the range of $\gamma \approx 1$. Brujan et al. observed that when laser-generated cavitation bubbles collapsed in
this region, the microjet travels through the bubble while the bubble is still collapsing, causing the microjet and the flow induced by the collapsing bubble to meet, resulting in a “splash”[18]. Figure 5.31e shows the distinct shape of the bubble at collapse for the standoff distance $\gamma = 1$, while Figure 5.31f shows how the collision of the flow of the collapsing bubble and the microjet results in a “splash” in the radial direction.

Figure 5.31: Volume fraction of the gas for $\gamma = 1.0$ at different times in the bubble formation and collapse
Figure 5.32, a series of pressure contour plots, shows that the pressure field behaves in a similar manner for $\gamma = 1.0$ as it did for $\gamma = 0.7$. As with bubble collapse for $\gamma = 0.7$, Figure 5.33 shows that the pressure wave generated by the second collapse has a much greater affect on the wall than the pressure generated by the first collapse. Figure 5.34, which shows the magnitude of the jet, exhibits microjet dynamics expected of bubble collapse. Although it is worth noting that the magnitude of the microjet is much higher for $\gamma = 1.0$ than for any of the other shorter standoff distances.

Figure 5.32: Contour plot of pressure for $\gamma = 1.0$ at different times in the bubble formation and collapse; note that pressure has been rescaled at each time step.
Figure 5.33: Change in pressure for CFD of bubble collapse at $\gamma = 1.0$ along the wall at different times and $x$ locations.

Figure 5.34: Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 1.0$ at the center of the bubble and as it travels towards the wall.

The final and largest standoff distance analyzed is $\gamma = 1.3$. Figure 5.35 shows that unlike the previous cases, for $\gamma = 1.3$ the bubble forms and collapses without touching the wall. It is only after the high-speed microjet travels from the center of the bubble to the wall that the bubble touches the wall. The collapse time for this simulation is approximately 190 $\mu$s, which of all of the different standoff distances discussed is the closest to the collapse time of a single bubble in a free field. This is because the bubble is far enough away from the wall that it initially behaves similar to the free bubble case.
Figure 5.35: Volume fraction of the gas for $\gamma = 1.3$ at different times in the bubble formation and collapse.

Figure 5.36 shows that the pressure profile in time is similar to the other bubble collapse cases, where the highest pressure is seen at the first and second bubble collapses.

Figure 5.37 shows the high pressure peaks generated by the bubble collapses, as with $\gamma = 0.5$ and $\gamma = 0.7$, the impact pressure on the wall at certain $x$ locations is greater at the second collapse than the first. Figure 5.38 shows a similar pattern for the jet velocity as previously seen for a standoff distance of $\gamma = 1.0$. 
Figure 5.36: Contour plot of pressure for $\gamma = 1.3$ at different times in the bubble formation and collapse; note that pressure has been rescaled at each time step.

(a) $t = 10 \mu s$
(b) $t = 100 \mu s$
(c) $t = 180 \mu s$
(d) $t = 210 \mu s$
(e) $t = 310 \mu s$
(f) $t = 400 \mu s$

Figure 5.37: Change in pressure for CFD of bubble collapse at $\gamma = 1.3$ along the wall at different times and $x$ locations.

(a) Pressure along the wall at different times
(b) Pressure along the wall at different $x$ locations
The CFD simulations of bubble collapse against a plate exhibited many of the same bubble dynamics seen in experimental research on the effect of standoff distance. Phillip and Lauterborn [17] found that the greatest amount of damage occurs at standoff distances less than 0.3 and between 1.2 and 1.4. Brujan et al. argued that for certain standoff distances, for example in the range between 0.5 and 1.1, the contact between the bubble and the wall decreases the amount of damage done to the wall [18]. The CFD simulation of bubble collapse at $\gamma = 0.2$ shows two very high pressure pulses, these intense pressure pulses combined with the close proximity of the bubble with the wall could account for high rates of cavitation damage at this standoff distance.

Figure 5.39a shows the maximum pressure calculated along the wall at the first and second bubble collapse for each standoff distance. As previously discussed, for $\gamma = 0.2$ the pressure pulse emitted at the first collapse is much greater than the pressure wave emitted upon the second bubble collapse. For the remaining standoff distances the pressure waves from both bubble collapses affect the wall similarly; emitting pressure waves of the same order of magnitude. Figure 5.39b shows the maximum pressure calculated at three different probe locations along the wall; since the pressure is saved at every time step for the probe locations it is expected that the maximum pressure over the entire length of the simulation might yield a slightly different result that the maximum
pressure at one specific time. This figure shows that the wall experiences high impact pressures at all of the standoff distances, most notably at $\gamma = 0.2$ which supports the conclusion that cavitation damage is more pervasive at this standoff distance because of the high impact pressures.

![Graphs showing maximum pressure calculated at the first and second collapse and maximum pressure after 100 $\mu$s at different $x$ locations along the wall.](image)

**Figure 5.39:** Maximum pressure calculated along the wall ($y = 0$) at different times and locations

The results of the CFD simulation for $\gamma = 1.3$ are not as straightforward to analyze. It appears that the magnitude of the jet velocity at the center of the bubble increases with increasing distance between the bubble and the plate, as can be seen in Figure 5.40. However the maximum jet velocity near the wall is higher for lower standoff distances, see Figures 5.22, 5.26 and 5.30. Philipp and Lauterborn [17] observed that jet impact velocity decreases for increasing standoff distances. They theorized that for $\gamma < 1$, since the bubble is in contact with the wall during the entire collapse, the impact jet velocity is equivalent to the maximum jet velocity [17]. As expected based on the results of Philipp and Lauterborn, the jet formation occurs earlier for bubble collapse at this standoff distance and is much smaller in magnitude by the time it reaches the wall. In fact, it becomes so small that Philipp and Lauterborn argue that at this standoff distance the jet does not contribute to cavitation damage [17].

An examination of the maximum impact pressure and the jet velocity accounts for why damage is so prevalent at $\gamma = 0.2$, but does not explain why there might be more
damage at $\gamma = 1.3$. In order to gain additional insight into how bubble collapse affects the wall, the wall shear stress was examined for each of the standoff distances. The time step where the maximum wall shear stress occurs coincides with the time step where the wall experiences the impulse pressure generated by the first collapse. For the majority of the different cases this occurs at $t = 210 \mu s$, as can be seen below in Figure 5.41a, a graph of the wall shear stress along the wall. Figure 5.41b shows the maximum wall shear stress calculated for each standoff distance. Dijkink and Ohl measured a wall shear stress of 3.5 kPa for a laser-generated bubble with $R_{\text{max}} = 0.75$ mm at $\gamma = 1.0$ [57]. The maximum wall shear stress measured by Dijkink and Ohl would appear to agree, at least from an order of magnitude comparison, with the maximum wall shear stress calculated from the CFD solutions. Figure 5.41b reveals that the largest maximum wall shear stress occurs at $\gamma = 1.2$, offering a possible explanation for why more damage is often observed at this standoff distance than at smaller standoff distances.
Figure 5.41: Wall shear stress calculated along the wall for different $\gamma$

The CFD simulations of bubble collapse at different standoff distances revealed very different bubble dynamics for each standoff distance, as expected based on both the experimental and computational literature on bubble collapse [17, 18, 19, 21, 22, 33, 36]. The CFD results showed good correlation between the trends of the magnitude of the microjet velocity and those of experimental work with laser-generated bubble collapse [17], as well as good correlation between trends shown in the computed impulsive pressures against the wall and measured experimental impulsive pressures [21]. There also seems to be good agreement between the wall shear stress calculated in the CFD simulations and experimental data of wall shear stress. This work agrees with the computational results of Johnsen and Colonius [25] who found that a homogeneous, compressible multiphase code could indeed capture the essential bubble dynamics of laser-generated bubble collapse, namely the microjet and the pressure pulses generated upon collapse.

### 5.8.2 Comparison with Experimental Data

Although the CFD results show trends that were comparable with experimental data, such as the formation of the microjet and the magnitude of the pressure pulses emitted upon collapse, up until this point no direct comparison has been made with experimental data. Philipp and Lauterborn measured the maximum jet velocity and the impact jet velocity of laser-generated cavitation bubbles at various standoff distances from a series
of high-speed photographs for a bubble with a maximum radius of $R_{\text{max}} = 1.45$ mm [17]. Table 5.9 shows the values of the maximum jet velocity, $v_{\text{max}}$, and the impact jet velocity, $v_{\text{imp}}$, for experimental data and CFD.

Table 5.9: Comparison of the maximum jet velocity and the impact jet velocity between CFD and experimental data for $\gamma = 0.7$ and $\gamma = 1.2$

<table>
<thead>
<tr>
<th>$\gamma$</th>
<th>$R_{\text{max}}$ (mm)</th>
<th>$v_{\text{max}}$ (m/s)</th>
<th>$v_{\text{imp}}$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experiment</td>
<td>CFD</td>
<td>Experiment</td>
</tr>
<tr>
<td>0.7</td>
<td>1.45</td>
<td>117</td>
<td>195.8</td>
</tr>
<tr>
<td>1.2</td>
<td>1.45</td>
<td>95.5</td>
<td>240.6</td>
</tr>
</tbody>
</table>

As can be seen from Table 5.9, the CFD results overpredict the experimental data, although the values are still on the same order of magnitude. Johnsen and Colonius et al. found a similar result in their computational analysis of bubble collapse, they theorized that in a laser-generated cavitation bubble the jet velocity could be sensitive to pulse width [25, 58]. The experimental results did not indicate the distance between the jet and the wall when the jet impact was measured, therefore a direct comparison could not be made between the CFD results and the experimental data. However, Figure 5.42 does seem to indicate that as the jet approaches the wall the magnitude of the jet is similar to the experimental data shown in Table 5.9.

![Figure 5.42: Magnitude of the velocity of the microjet for bubble collapse at $\gamma = 0.7$ and $\gamma = 1.2$ as the jet travels from the center of the bubble towards the wall](image-url)

(a) Magnitude of velocity at $x = 0$ along $y$ at different times for $\gamma = 0.7$

(b) Magnitude of velocity at $x = 0$ along $y$ at different times for $\gamma = 1.2$
Although a direct comparison could not be made between the experimental work of Tomita and Shima [21] because different bubble sizes and standoff distances were examined, Table 5.10 shows a comparison between the impulse pressures measured experimentally and impulse pressures calculated from the CFD results. Although the bubble generated by Tomita and Shima is slightly larger, the same trends can be seen in both the experimental and computational results. The maximum impact pressure is the highest for small standoff distances, where the bubble collapses close to the wall, and decreases slightly with increasing standoff distance.

Table 5.10: Comparison of the maximum impact pressure between CFD and experimental data for different standoff distances

<table>
<thead>
<tr>
<th>$\gamma$</th>
<th>$R_{\text{max}}$ (mm)</th>
<th>$p_{\text{max}}$ (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experiment</td>
<td>CFD</td>
</tr>
<tr>
<td>0.17</td>
<td>3.5</td>
<td>21.1</td>
</tr>
<tr>
<td>0.2</td>
<td>1</td>
<td>-</td>
</tr>
<tr>
<td>0.56</td>
<td>3.5</td>
<td>11.9</td>
</tr>
<tr>
<td>0.5</td>
<td>1</td>
<td>-</td>
</tr>
<tr>
<td>0.7</td>
<td>3.5</td>
<td>7.1</td>
</tr>
<tr>
<td>0.7</td>
<td>1</td>
<td>-</td>
</tr>
<tr>
<td>1.0</td>
<td>3.5</td>
<td>9.5</td>
</tr>
<tr>
<td>1.0</td>
<td>1</td>
<td>-</td>
</tr>
<tr>
<td>1.44</td>
<td>3.5</td>
<td>8.0</td>
</tr>
<tr>
<td>1.3</td>
<td>1</td>
<td>-</td>
</tr>
</tbody>
</table>

5.8.3 Effect of Gravity on Bubble Collapse

The effect of gravity was analyzed for bubble collapse when $\gamma = 1.3$, comparing both the solution with gravity and with gravity neglected for both bubble collapse above and under a plate. Gravity appeared to have an insignificant affect on the solution of the velocity field, there was a negligible difference between the shape and the magnitude of the microjet. The volume fraction of the gas and the pressure fields however, were affected by gravity, especially as the solution advanced in time.
Figure 5.43: Volume fraction of the gas versus time for bubble collapse above and under a wall for $\gamma = 1.3$ at $x = 0.1$ mm and $y = 0$ mm

Figure 5.43 shows the volume fraction of the gas versus time for the CFD solution of bubble collapse at $\gamma = 1.3$ when gravity is neglected, as well as when the effects of gravity are included. The same location at the wall is examined in both Figure 5.43a and 5.43b for bubble collapse above a wall and bubble collapse under a wall respectively. Since $\gamma = 1.3$, it is not until after the first collapse that the bubble reaches the wall, which is why $\alpha_1$ is initially zero. Figure 5.43 shows that the effect of gravity on $\alpha_1$ become significant around 450 $\mu$s; for the case of bubble collapse above a wall gravity increases $\alpha_1$, while for the case of bubble collapse under a wall including gravity decreases $\alpha_1$. Similarly Figure 5.44, a plot of pressure at the same location at the wall, shows that the pressure pulse resulting from the second bubble collapse is higher when gravity is included in the simulation for bubble collapse above a wall, but does not significantly affect the pressure for bubble collapse under a wall.
Figure 5.44: Pressure versus time for bubble collapse above and under a wall for $\gamma = 1.3$ at $x = 0.1$ mm and $y = 0$ mm.

Although a more detailed examination of the effects of gravity on bubble collapse is needed before any significant conclusion can be drawn, this preliminary analysis seems to agree with the results of Chan et al. [34], who found that the impact pressure was higher when the bubble was collapsed above a plate than when it was collapsed under a plate.

5.8.4 Grid Refinement Study

A grid refinement study similar to the one conducted for single bubble collapse in a free field was conducted on the mesh used in the CFD simulations of bubble collapse against a wall. The mesh study was conducted for bubble collapse at a standoff distance of $\gamma = 1$. The CFD solution was compared for three different grids; where the grid used in sections 5.8.1, 5.8.2 and 5.8.3 was the coarsest mesh examined. Figure 5.45 shows the pressure at the position $y = s$, the initial center of the bubble, as a function of $x$ for the three different meshes. The pressure was calculated at three points in time; the first just before the bubble reaches its maximum radius, the second at the maximum radius of the bubble and the third just after the bubble begins to collapse. As in section 5.7.3, the greatest difference between the pressure values of each mesh occurs at the start of the bubble collapse. Figure 5.45a shows a negligible difference in the pressure field for each of the three meshes before the bubble collapse. While Figures 5.45b and 5.45c
show a slight disparity between the different meshes. Despite these small discrepancies, it appears that the mesh used in the CFD simulations of bubble collapse against a wall was refined appropriately.

![Graphs showing pressure versus x at y = s at different times and meshes](image)

Figure 5.45: Pressure versus x at y = s at three different times and three different meshes

### 5.9 Discussion

A homogeneous multiphase compressible finite volume method was used to study the formation and collapse of a laser-generated cavitation bubble. The CFD solution of a cavitation bubble in a free field was comparable to both the solution of the Rayleigh-Plesset equation and experimental data. The radius of the bubble during its initial expansion and collapse was well correlated with the radius determined from the solution...
of the Rayleigh-Plesset equation and experimental data. However, after the first collapse the CFD solution predicted a radius for the rebound bubble much larger than the radius seen in experimental data. This discrepancy could be because the CFD model used does not include a phase change model and cannot accurately model the temperature change across the vapor-liquid boundary. The bubble dynamics of collapse against a wall were then investigated at different standoff distances. The standoff distance was varied in order to model the different bubble behaviors that occur at different distances to the wall. The compressible CFD solver was able to capture both the microjet and the pressure pulse emitted upon the collapse of the bubble. The magnitude of the microjet and the impact pressure on the wall were then compared with experimental data. It appears that the compressible multiphase solver was able to successfully capture the dynamics of bubble collapse against a wall.
6

Summary

6.1 Summary

Cavitation occurs in flowing fluids when a liquid experiences a decrease in pressure below the vapor pressure of the liquid, causing vapor cavities to form. Cavitation erosion occurs when these small bubbles collapse and impinge against solid boundaries; leading to damage of the surface. Cavitation affects a wide range of industries, and can affect hydraulic machinery by damaging impeller blades and valves. Cavitation can affect machinery that is continuously underwater, such as ship propeller blades and turbomachinery submerged in water. Bubble collapse against a surface over time will lead to large areas of pitting. These pits will gradually cause an erosion of the material, leading to an increase in the turbulence of the flow as well as additional nucleation sites, which in turn results in an increase in cavitation bubbles.

Research regarding cavitation erosion has focused either on empirical correlations that can be used to predict cavitation or on understanding the dynamics of single bubble collapse. A significant portion of the experimental work performed on single bubble dynamics focuses on using a laser to generate a bubble that can be collapsed against a surface. The goal of this work was to design a simple experiment using a Nd:YAG laser to generate a bubble, and to then model that experiment using CFD. The experimental portion of this work focused on creating an experimental setup, similar to those of previous researchers, where a single bubble was created by expanding and then focusing
the beam of a laser into a tank of water.

The formation and collapse of a laser-generated cavitation bubble was then modeled computationally in the open source C++ library OpenFOAM. The Rayleigh-Plesset equation was first used to find an appropriate approximation of the bubble growth as a function of time. A compressible flow model was then used to find the pressure and velocity fields of the growing and collapsing bubble, using the compressible conservation of mass and conservation of momentum equations. In addition, the energy equation was solved separately in order to give a temperature distribution for the formation of a single laser-generated cavitation bubble. After successfully modeling both the heating of the water by the laser and the collapse of a bubble near a wall, future work will focus on creating a CFD model that can simulate the complete formation and collapse of a laser-generated cavitation bubble. This computational method would solve the mass, momentum and energy equations simultaneously, and would include a phase change model that would account for the micro-scale evaporation of the temperature as a laser-generated bubble forms. Such a computational model might better capture the dynamics of a single laser-generated bubble formed in a free field. Although the computational results discussed in this thesis relied on a CFD method that only solved the mass and momentum equations, the complex dynamics of bubble collapse against a wall were accurately captured using a homogeneous multiphase compressible finite volume method.

6.2 Conclusions

The thermal analysis completed in Chapter 4 confirmed that a source term could be added to the thermal energy equation that would account for the heating of the water by a Nd:YAG laser. The source term was dependent on the energy output of the laser, the reflectivity and absorption of the water, the radius of the laser beam, the laser pulse and the location of the beam in the water. A simple experiment was created where a laser bubble was generated using a Nd:YAG laser. That experimental setup was then modeled in the CFD simulation of the heating of the water by the laser. The resulting computational solution revealed that the temperature profile mirrored the
Gaussian beam profile, with the temperature at the focal volume rapidly increasing and decreasing in a Gaussian shape. The temperature generated by the laser exceeded 373 K, the boiling point of water. The model was able to simulate the temperature rise by a laser that leads to the formation of a vapor cavity.

Chapter 3 focused on the solution of the Rayleigh-Plesset equation. A derivation of the Rayleigh-Plesset equation was included along with a detailed explanation of the assumptions that were made in the derivation. The effect of the initial conditions on the solution of the Rayleigh-Plesset equation was examined, including the effect of the pressure ratio on the solution. The pressure ratio was found to have a significant affect on both the maximum bubble size and the collapse time. As the compression ratio for a laser-generated cavitation bubble is still not precisely known, a series of compression ratios were examined using the Rayleigh-Plesset equation in order to determine the corresponding pressure ratios. It was found that the pressure ratio increases exponentially with an increase in the compression ratio. This means that although an actual laser-generated cavitation bubble may have a compression ratio of between 20 and 70, for practical purposes a compression ratio of 10 leads to a much smaller pressure ratio that is easier to accurately model using CFD.

The CFD solutions for bubble collapse using a homogeneous multiphase compressible model were discussed in great detail in Chapter 5. CFD simulations were completed for both single bubble collapse in a free field and single bubble collapse against a wall. The results for the single bubble collapse in a free field were then compared to the solution of the Rayleigh-Plesset equation. Although the Rayleigh-Plesset equation is a simplified model for bubble dynamics in an infinite medium, the correlation was good between the Rayleigh-Plesset equation and the CFD simulation for the initial bubble formation and collapse. A comparison of the bubble radius during expansion and collapse was made between the CFD solution and experimental data of a laser-generated cavitation bubble. There was exceptionally good agreement between the CFD results and the experimental data during the initial formation and collapse. However, the CFD simulation overpredicted the size of the rebound bubble that occurs after the first collapse. Several possible explanations exist, the most probable is that the rebound bubble is larger because the
CFD model does not include a phase change model, nor does it account for mass and heat transfer between the liquid-vapor interface.

In addition to CFD simulations of laser bubble formation and collapse in a free field, the collapse of a bubble against a wall was also examined. The standoff distance, a parameter measuring the distance from the center of the initial bubble to the wall, was varied in order to examine the different bubble dynamics that are produced by various proximities to the wall. All of the CFD simulations modeled the basic dynamics of bubble collapse; the bubble expands, quickly collapses and a microjet forms that travels from the center of the bubble towards the wall. In addition to the microjet, an intense pressure pulse is also emitted upon the collapse of the bubble. The generation of a high speed microjet and a pressure wave upon the collapse of a cavitation bubble are thought to contribute to cavitation damage. The results of the bubble collapse simulations were then compared to experimental data for laser-generated bubble collapse against a plate. The microjet velocity computed in the CFD simulations was slightly higher than the microjet velocity measured experimentally. Although a direct comparison could not be made between the impact pressure computed in the CFD results and experimental pressure measurements, the simulated impulse pressure was on the same order of magnitude as the experimental impact pressure. The trends observed in the velocity and pressure profiles of the CFD simulations were very similar to those observed in the experimental data. Overall there seemed to be a good agreement between the CFD results and the experimental data of laser-generated bubble collapse against a wall.

6.3 Future Work

The compressible flow solver used to model bubble collapse incorporated the mass and momentum equations as well as two equations of state. This solver uses a pressure-based finite volume method; using a pseudo-fluid approach to solve only one set of transport equations in addition to a volume fraction equation that accounts for the differences in the densities of the two phases. The CFD and the Rayleigh-Plesset equation showed remarkable similarity in their prediction of the radius of the bubble up until the first
bubble collapse. A thermal analysis was also completed that modeled the temperature change created by a laser in a tank of water; simulating the temperature rise that occurs when a cavitation bubble is generated by a laser.

The compressible two-phase model described in section 2.3.2 combined with the energy equation analysis described in section 2.2.4 would result in a flow solver which includes the compressible mass, momentum and energy equations. The compressible CFD model used in this analysis to model bubble collapse was a homogeneous mixture model, a model which assumes that the temperature and the velocity of the gas are equal to the temperature and velocity of the liquid at the liquid-vapor interface. This assumption eliminates the need for two sets of transport equations and is critical to the pseudo-fluid approach. For the case of a laser-generated cavitation bubble, however, where a bubble is being created by a huge and rapid temperature increase, it is no longer acceptable to make the assumption that the two phases are in thermal equilibrium at the liquid-vapor interface. Therefore, it is necessary to include a phase change model which accounts for the temperature differences between the two phases at the interface without having to solve two sets of transport equations. Such a model would provide a CFD solution for the formation and collapse of a laser-generated cavitation bubble that includes a coupled solution for the temperature, pressure and velocity fields. As with the other models, the goal would be to implement it into OpenFOAM. This model would be a powerful tool in understanding the physics of single bubble formation and collapse, as well as cavitation damage.

The initial goal of this work was to develop a model that solved the mass, momentum and energy equations; however, implementing a phase change model, as well as the heat and mass transfer across the two phases, requires complex inter-facial dynamics. The end result of this work, therefore, was modeling the temperature field separate from the pressure and velocity fields, instead of in a single coupled solver. Future work would involve finishing the model that was initially attempted; implementing the energy equation and the phase change model into the compressible flow solver described in detail in section 2.3.2. That model could then be used to computationally model various experimental work involving cavitation erosion. Although the dynamics of bubble collapse
can be modeled using the CFD method outlined in this thesis, adding the energy equation and a phase change model to the compressible multiphase flow solver would allow for a more accurate analysis of the formation of a laser-generated cavitation bubble. In addition to the benefit of accurately modeling both the formation and collapse of a laser-generated cavitation bubble, such a model would lead to a much better prediction of the rebound bubble formed after the first bubble collapse. Furthermore, such a model would eliminate much of the uncertainty surrounding the initial conditions in CFD simulations of bubble collapse. Currently when compressible homogeneous multiphase methods are used to model CFD bubble collapse, the simulation is either started at the maximum radius, where the initial conditions are approximated using analytical equations, or started at an approximate initial bubble radius where the initial conditions are determined from the solution of the Rayleigh-Plesset equation.

6.4 Discussion

Cavitation is a complex flow phenomenon that occurs when the pressure of a fluid drops below the vapor pressure, causing a cavity to form. In an attempt to understand the process of cavitation bubble collapse, and its affect on solid surfaces, this analysis focused on the formation and collapse of laser-generated cavitation bubbles. After devising an experiment where a cavitation bubble was formed using a Nd:YAG laser, a thermal analysis of that experimental setup was conducted in OpenFOAM. This analysis confirmed that the inclusion of a source term in the energy equation would result in a temperature profile where a small area of the liquid was heated past its boiling point, resulting in the formation of a vapor bubble. A compressible multiphase flow model was then used to model the formation and collapse of a single cavitation bubble in both a free field and against a solid surface. A good correlation was found between the CFD model, the Rayleigh-Plesset equation and experimental data during the initial formation and collapse of a single bubble in a free field. The size of the rebound bubble however, was overpredicted by the CFD solver. This could be because the CFD model did not account for phase transition, or mass and heat transfer between the vapor-liquid interface.
Despite these limitations, the compressible flow model was able to accurately capture the essential flow dynamics of bubble collapse against a wall, namely the formation of a microjet and the intense pressure pulse that are both initiated at the moment of collapse.
Appendix

Code Modifications

A.1 Modifications to scalarTransportFoam

```
/*--------------------------------*- C++ -*----------------------------------*
 \file    F i e l d
 \author  O p e nFOAM: The Open Source CFD Toolbox
 \author  Copyright held by original author
 \author  M a n i p u l a t i o n

License
This file is part of OpenFOAM.

OpenFOAM is free software; you can redistribute it and/or modify it
under the terms of the GNU General Public License as published by the
Free Software Foundation; either version 2 of the License, or (at your
option) any later version.

OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License
for more details.

You should have received a copy of the GNU General Public License
along with OpenFOAM; if not, write to the Free Software Foundation,
Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA

Application
 scalarTransportFoam

Description
Solves a transport equation for a passive scalar

\*-------------------------------------*/

#include "fvCFD.H"
```
int main(int argc, char *argv[])
{
    # include "setRootCase.H"
    # include "createTime.H"
    # include "createMesh.H"
    # include "createFields.H"
    # include "readLaserHeatingDict.H"

    Info<< "Calculating scalar transport\n" << endl;
    dimensionedScalar twoTimeA("twoTimeA",dimTime,2.*timeA.value());
    const scalar pi (M_PI);
    dimensionedScalar zero("zero",dimTemperature/dimTime,0.0);
    dimensionedScalar fLength1("fLength1", dimLength, 0.0);
    dimensionedScalar fLength2("fLength2", dimLength, 0.0);
    # include "CourantNo.H"

    while (runTime.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;
        // input the time
        tau += runTime.deltaT();
        # include "readSIMPLEControls.H"

        // calculate the radius from the x and y components
        radius = sqrt(sqr(mesh.C().component(1))+sqr(mesh.C().component(2)));
        // convert fLength , a dimensionedScalar to xLength , a volScalarField
        xLength = fLength;
        // calculate f1 and f2 where wfLaser is the beam radius at the focal volume
        // and lambda is the wavelength of the laser
        fLength1 = fLength - (3000)*(sqr(wfLaser)/lambda);
        fLength2 = fLength + (3000)*(sqr(wfLaser)/lambda);
        // convert fLength1 and fLength2 , both dimensionedScalars , to volScalarField
        xLength1 = fLength1;
        xLength2 = fLength2;
        // Convert DT1, cp1 and rho1 from dimensionedScalar to volScalarField
        // Now they can be outputted and double checked. Note this is not essential.
        DT = DT1;
        cp = cp1;
        rho = rho1;
        xpos = mesh.C().component(0);
        ypos = mesh.C().component(1);
        zpos = mesh.C().component(2);
        rLengthR = rLength;
        time = tau;

        // Calculate the beam radius at every x location
forAll(wXLaser, cellI)
{
    if (xpos[cellI] < xLength1[cellI]) {
        wXLaser[cellI] = ((lambda.value()*(fLength.value()-xpos[cellI]))
        /(pi*wfLaser.value()));
    } else if ((xpos[cellI] >= xLength1[cellI]) && (xpos[cellI] < xLength[cellI]) {
        wXLaser[cellI] = (wfLaser.value()*Foam::sqrt(1.0 +
        sqr((fLength.value()-xpos[cellI])/rLength.value())));
    } else if ((xpos[cellI] >= xLength[cellI]) && (xpos[cellI] < xLength2[cellI]) {
        wXLaser[cellI] = (wfLaser.value()*Foam::sqrt(1.0 +
        sqr((xpos[cellI]-fLength.value())/rLength.value())));
    } else {
        wXLaser[cellI] = ((lambda.value()*(xpos[cellI]-fLength.value()))
        /(pi*wfLaser.value()));
    }
}
// calculate the beam radius at the boundary conditions for every x location
forAll(wXLaser.boundaryField(), patchI)
{
    if (xpos[patchI] < xLength1[patchI]) {
        wXLaser.boundaryField()[patchI] = ((lambda.value()*(fLength.value()-xpos[patchI]))
        /(pi*wfLaser.value()));
    } else if ((xpos[patchI] >= xLength1[patchI]) && (xpos[patchI] < xLength[patchI]) {
        wXLaser.boundaryField()[patchI] = (wfLaser.value()*Foam::sqrt(1.0 +
        sqr((fLength.value()-xpos[patchI])/rLength.value())));
    } else if ((xpos[patchI] >= xLength[patchI]) && (xpos[patchI] < xLength2[patchI]) {
        wXLaser.boundaryField()[patchI] = (wfLaser.value()*Foam::sqrt(1.0 +
        sqr((xpos[patchI]-fLength.value())/rLength.value())));
    } else {
        wXLaser.boundaryField()[patchI] = ((lambda.value()*(xpos[patchI]-fLength.value()))
        /(pi*wfLaser.value()));
    }
}
// calculate the spatial profile, the temporal profile and the irradiance
// of the laser in order to double check the solution
sProfileZ = exp(-alpha*mesh.C().component(0));
sProfileR = exp(-2.0*(sqr(radius/wXLaser)));
irrad = (pLaser/(0.0139*pi*sqr(wXLaser)));

// calculate the source term => pLaser is the laser power, wXLaser is the beam
// radius, rLaser is the reflectivity of the water, alpha is the absorption
// coefficient, mesh.C().component(0) is x, and timeA is the rise time of the
// laser intensity and twoTimeA is twice the rise time of the laser intensity
if (tau.value() < timeA.value()) {
    source = (1/(rho*cp))*(pLaser/((0.0139)*pi*sqr(wXLaser)))*(1-rLaser)*alpha
}
*exp(-2.0*sqrt(radius/wXLaser))*exp(-alpha*mesh.C().component(0))*
(tau.value()/timeA.value());
}

else if ( (tau.value() >= timeA.value()) && (tau.value() < twoTimeA.value()) ) {
    source = (1/(rho*cp))*(pLaser/((0.0139)*pi*sqr(wXLaser)))*(1-rLaser)*alpha
*exp(-2.0*sqrt(radius/wXLaser))*exp(-alpha*mesh.C().component(0))
*(twoTimeA.value()-tau.value())/timeA.value();
}

else {
    source = zero;
}

for (int nonOrth=0; nonOrth<=nNonOrthCorr; nonOrth++)
{
    solve
    {
        fvm::ddt(T) + fvm::div(phi, T) - fvm::laplacian(DT,T) = source;
    }
}

runTime.write();

Info<< "End\n" << endl;

return 0;
}

// ############################################################################///
Bibliography


117


