Numerical Simulation of Airlift Pumps Operating under Two-Phase Flow Conditions for Aquaculture Systems

by

Mohammad Shallouf

A Thesis
presented to
The University of Guelph

In partial fulfilment of the requirements for the degree of
Master of Applied Science
in
Engineering

Guelph, Ontario, Canada
© Mohammad Shallouf, January 2019
ABSTRACT

NUMERICAL SIMULATION OF AIRLIFT PUMPS OPERATING UNDER TWO-PHASE FLOW CONDITIONS FOR AQUACULTURE SYSTEMS

Mohammad Shallouf
University of Guelph, 2018
Advisor: Dr. Wael Ahmed

The airlift pump has been around since the late 18th century, yet it is rarely seen or used in industry. Recently, the aquaculture industry has shown great potential for the use of airlift pumps operating under two-phase flow conditions to increase the operating efficiency of such facilities. This thesis presents both numerical and experimental investigations of the operation and performance characteristics of an airlift pump system. Experimental verification was also performed of an airlift pump that was integrated with a fish farm raceway in an aquaculture facility. Utilizing the Volume of Fluid (VOF) multiphase model along with the K-ε turbulence model, a numerical study using ANSYS Fluent 19.0 was performed for the two-phase flow hydrodynamics in airlift pump injectors. The study was carried out for both the two-phase flow in the pump riser, and for the flow in an open channel raceway. Also, the performance of the airlift pump was evaluated experimentally for a 2.54 cm diameter airlift pump in the lab. The numerical results were found to be in agreement with the experiments within ±20%. These simulation results were used to scale an airlift pump to operate in a 3.5 m³ raceway system. The numerical results had an average RMS agreement of ±15% with the experimental results. The present study was found to present a great tool for optimizing the airlift pump performance, as well as modifying the design of future aquaculture systems or other two-phase flow applications.
DEDICATION

Leem and Ahmad
Fatima and Mohamad
Athla and Issaf
ACKNOWLEDGEMENTS

I would like to sincerely thank my advisor, Dr. Wael Ahmed for his support, guidance and endless patience with me throughout this journey. I would also like to deeply thank Dr. Sherif Abdou for the constant support, insight and helpful tips and tricks that got me through the mazes of simulation work.

To all my family, thank you for always being there when I needed it, for being my rock and foundation when I was shaky and for always loving me even during the times when I wasn’t very lovable.

A big thank you to all the friends and people I met and worked with at Gryph Energy lab. You all made this experience much more memorable and immensely more enjoyable. A special thank you to Thariq Mohammed, Shaker Bukhari, Jeremy Chau, Adib Fatayerji, Joshua Rosettani, Rashal Abed, Domenica De Nicola for all the help, support, memories and most importantly laughs that we all shared.
# TABLE OF CONTENTS

Abstract .......................................................................................................................... ii
Dedication ....................................................................................................................... iii
Acknowledgements ....................................................................................................... iv
Table of Contents ......................................................................................................... v
List of Tables ................................................................................................................ viii
List of Figures ............................................................................................................... ix
Nomenclature ............................................................................................................... xii

Chapter 1 - Introduction ............................................................................................. 1
  1.1 Thesis Objectives ................................................................................................. 3
  1.2 Thesis Outline ...................................................................................................... 3

Chapter 2 - Literature Review ...................................................................................... 4
  2.1 Aquaculture Systems and Engineering ............................................................... 4
    2.1.1 Open-Cage Systems ...................................................................................... 5
    2.1.2 Flow-Through Systems ............................................................................... 8
    2.1.3 Recirculating Aquaculture Systems (RAS) .................................................. 10
  2.2 Airlift Pumps Under Two-Phase Conditions ..................................................... 14
    2.2.1 Theory of Operation ..................................................................................... 16
    2.2.2 The Use of Airlift Pumps in Aquaculture Systems ........................................ 21
  2.3 Two-Phase Flow Numerical Modelling Using CFD ........................................... 23
    2.3.1 Numerical Modelling of Bubble Columns and Airlift Reactors ............... 23
    2.3.2 Numerical Modelling of Two-Phase Flow in Vertical Pipes ................... 31
    2.3.3 Numerical Modelling of Free Surface Flow ............................................. 37
2.4 The Use of CFD in Designing Aquaculture Systems ................................................. 38
2.5 Gaps in the Literature ............................................................................................... 40
2.6 Study Objectives ........................................................................................................ 40

Chapter 3 - Experimental Analysis ................................................................................ 42
3.1 Laboratory Setup ....................................................................................................... 42
3.2 Field Setup ............................................................................................................... 45
3.3 Instrumentation ......................................................................................................... 49
  3.3.1 Air Flow Meter .................................................................................................. 49
  3.3.2 Graduated Measurement Tank ........................................................................ 49
  3.3.3 Capacitance Sensor .......................................................................................... 49
  3.3.4 Acoustic Doppler Velocimeter (ADV) ........................................................... 50
3.4 Calibration and Uncertainty ...................................................................................... 51

Chapter 4 - Numerical Analysis ..................................................................................... 53
4.1 Airlift Injector in Single Phase Conditions ............................................................ 53
4.2 Airlift Pump Modelling in Two-Phase Flow Conditions .......................................... 55
  4.2.1 Volume of Fluid Model (VOF) ....................................................................... 60
4.3 Turbulence Parameters ............................................................................................. 62
  4.3.1 Reynolds Averaged Navier-Stokes Equations ................................................ 63
4.4 Geometry and Meshing ............................................................................................. 67
  4.4.1 Geometry and Boundary Conditions ............................................................... 67
  4.4.2 Meshing ............................................................................................................ 75
4.5 Discretization and Solution Method .......................................................................... 81
4.6 Summary of Assumptions ......................................................................................... 85

Chapter 5 - Results and Discussion ............................................................................... 86
5.1 Mesh Independence Study ........................................................................................................... 86
  5.1.1 Dual Injection Airlift ............................................................................................................ 86
  5.1.2 Airlift Pump System ............................................................................................................ 90
5.2 Single-Phase Flow in The Dual Injection Airlift ........................................................................ 92
  5.2.1 Velocity Magnitude ............................................................................................................ 95
  5.2.2 Vorticity ............................................................................................................................ 99
  5.2.3 Turbulent Kinetic Energy .................................................................................................. 101
  5.2.4 Turbulence Intensity .......................................................................................................... 103
  5.2.5 Static Pressure .................................................................................................................. 105
5.3 Two-Phase Flow in an Airlift Pump System ............................................................................ 110
  5.3.1 Void Fraction ...................................................................................................................... 112
  5.3.2 Velocity ............................................................................................................................ 121
  5.3.3 Pressure ............................................................................................................................ 128
  5.3.4 Performance Curve ........................................................................................................... 129
5.4 Flow in an Aquaculture Raceway .......................................................................................... 136

Chapter 6 - Conclusion ................................................................................................................ 140

References ...................................................................................................................................... 142

APPENDIX A: Experimental Data ................................................................................................. 154

APPENDIX B: Computational Data ............................................................................................... 156
LIST OF TABLES

Table 2-1-Summary of Bubble Column Literature ............................................................... 29
Table 2-2- Summary of Vertical Two-Phase Flow Literature ............................................. 35
Table 3-1- Summary of Instrumentation Uncertainties ....................................................... 52
Table 4-1- Summary of Multi-Phase CFD Models ............................................................... 59
Table 4-2- Summary of Meshing ......................................................................................... 81
Table 4-3- Differencing Schemes Used .............................................................................. 84
Table 5-1- Summary of Boundary Conditions – Airlift Injector Mesh Independence Study ................................................................................................................................. 87
Table 5-2- Summary of Boundary Conditions – Airlift Pump System Mesh Independence Study ................................................................................................................................. 90
Table 5-3- Fluid Properties Used in the Work ................................................................. 92
Table 5-4- Summary of Boundary Conditions- Dual Injection Airlift Pump ................. 93
Table 5-5- Summary of Solution Parameters Monitored – Dual Injector Airlift Pump .... 94
Table 5-6- Summary of Boundary Conditions – Airlift Pump System ....................... 110
Table 5-7- Summary of Solution Parameters Monitored – Airlift Pump System ....... 111
Table 5-8-Summary of Boundary Conditions - Raceway .............................................. 136
Table 5-9- Summary of Solution Parameters Monitored - Raceway ......................... 137
Table 6-1- Experimental Pressure Measurements - Air ................................................ 154
Table 6-2- Experimental Data for 2.54 Airlift Pump System ........................................ 154
Table 6-3- Experimental Raceway Velocities .................................................................. 155
Table 6-4- Computational Pressure Monitors- Air ........................................................ 156
Table 6-5- Computational Pressure Monitor - Water ....................................................... 156
Table 6-6- Computational Performance Data ................................................................... 157
Table 6-7- Computational Raceway Velocities ................................................................. 158
LIST OF FIGURES

Figure 2-1- Open Cage Systems in Norway [12] (Top Left), Scotland [13] (Top Right) and Schematic (Bottom) .......................................................................................................................... 6

Figure 2-2- Flow-Through System Schematic (Top) and Operational Flow-Through Systems in the US (Bottom) [18]–[21] ........................................................................................................................................... 9

Figure 2-3- RAS Schematic (Top) and Operational RAS Around the World (Bottom) [24]–[27] .................................................................................................................................................... 12

Figure 2-4- Types of Airlift Pump Systems: Ordinary (Left), Step (Center), Tapered (Right) ...................................................................................................................................................... 15

Figure 2-5- Airlift Pump System Schematic .................................................................................................................. 17

Figure 2-6- Two-Phase Flow Patterns .............................................................................................................................. 18

Figure 2-7- Airlift Pump for Transporting Fish ................................................................................................................. 22

Figure 2-8- Bubble Column (Left), Airlift Reactor (Right) Schematics .......................................................................... 24

Figure 2-9- Schematic of Components of this Work’s Objective ..................................................................................... 41

Figure 3-1- Experimental Setup ...................................................................................................................................... 44

Figure 3-2- Aquaculture Airlift Pump System .................................................................................................................. 45

Figure 3-3- Installation of the Airlift Pump System onto the Raceway ............................................................................ 46

Figure 3-4- Velocity Measurements Using ADV ............................................................................................................. 47

Figure 3-5- Experimental Velocity Measurement Schematic ...................................................................................................... 48

Figure 3-6- SonTek Acoustic Doppler Velocimeter [136]....................................................................................................... 51

Figure 4-1- Airlift Pump Injector Designs as Described by Ahmed et al. [68] ................................................................. 68

Figure 4-2- Dual Injection Airlift Pump Fluid Domain and Dimensions ..................................................................................... 69

Figure 4-3- Boundary Conditions for the Dual Injector Airlift .......................................................................................... 70

Figure 4-4- Airlift Pump System Domain and Dimensions .................................................................................................. 71

Figure 4-5- Boundary Conditions for the Airlift Pump System .......................................................................................... 72
Figure 4-6- Aquaculture Raceway Model ................................................................. 73
Figure 4-7- Raceway Domain Dimensions .............................................................. 74
Figure 4-8- Boundary Conditions for the Raceway Domain .................................... 75
Figure 4-9- Divided Dual Injector Domain ............................................................ 77
Figure 4-10- Dual Injector Airlift Pump System Meshing ..................................... 79
Figure 4-11- Raceway Domain Mesh: Side View (Top), Front View (Bottom) ......... 80
Figure 4-12- SIMPLE Discretization Algorithm .................................................... 83
Figure 5-1- Contour Plots of (a) Static Pressure and (b) Velocity ......................... 89
Figure 5-2- Mesh Independence Performance Results ......................................... 91
Figure 5-3- Velocity Magnitude Contour Plot: (a) Air, (b) Water ....................... 96
Figure 5-4- Air Vs. Water Radial Velocity Profile ............................................... 98
Figure 5-5- Vorticity Magnitude Contour Plot: (a) Air, (b) Water ....................... 100
Figure 5-6- Turbulence Kinetic Energy Contour Plot: (a) Air, (b) Water ............ 102
Figure 5-7- Turbulence Intensity Contour Plot: (a) Air, (b) Water ..................... 104
Figure 5-8- Static Pressure Contour Plot: (a) Air, (b) Water .............................. 106
Figure 5-9- Dual Injector Air Phase Pressure Drop ............................................. 107
Figure 5-10- Measured Vs. Predicted Pressure Drop .......................................... 108
Figure 5-11- Dual Injector Water Phase Pressure Drop ....................................... 109
Figure 5-12- Void Fraction Time Sequence at $zd = 15$: (a) Numerical Simulation (b) Experimental ................................................................. 114
Figure 5-13- Void Fraction in Airlift Pump System Time Sequence ..................... 116
Figure 5-14- Time Averaged Radial Void Fraction Profile at $mair = 2.57 kgh$ ........ 118
Figure 5-15- Void Fraction Vs. Air Mass Flow Rate .......................................... 120
Figure 5-16- Time Average Radial Velocity Profile at $mair = 2.57 kgh$ ............... 122
Figure 5-17 - Velocity Contour at the Airlift Injector ................................................. 124
Figure 5-18 - Velocity Vector at $zd = 15$ and Airlift Injector ........................................ 125
Figure 5-19 - Area Weighted, Time Average Velocity Magnitude Vs. Air Flow Rate .... 127
Figure 5-20 - Area Weighted, Time Averaged Static Pressure Vs. Air Flow Rate ...... 128
Figure 5-21 - Airlift Pump Performance Curve ............................................................... 130
Figure 5-22 - Measured Vs. Predicted Water Flow Rate .................................................. 131
Figure 5-23 - Taitel et al. [64] Flow Pattern Map ........................................................... 134
Figure 5-24 - Raceway Domain Measurement Schematic ................................................. 137
Figure 5-25 - X-Direction Velocity Profile in Raceway .................................................... 138
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d$</td>
<td>Diameter</td>
<td>$Sr$</td>
<td>Submergence Ratio</td>
</tr>
<tr>
<td>$E$</td>
<td>Energy</td>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
<td>$v$</td>
<td>Velocity</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravity</td>
<td>$\alpha$</td>
<td>Void Fraction</td>
</tr>
<tr>
<td>$h$</td>
<td>Enthalpy</td>
<td>$\alpha^*$</td>
<td>Turbulence Viscosity factor</td>
</tr>
<tr>
<td>$J$</td>
<td>Diffusion Flux</td>
<td>$\beta$</td>
<td>Slope Limiter</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent Kinetic Energy</td>
<td>$\Gamma$</td>
<td>Diffusivity</td>
</tr>
<tr>
<td>$K_{eff}$</td>
<td>Effective Conductivity</td>
<td>$\mu$</td>
<td>Dynamic Viscosity</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass Flow Rate</td>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>$\sigma$</td>
<td>Prandtl Number</td>
</tr>
<tr>
<td>$Q$</td>
<td>Volumetric Flow rate</td>
<td>$\tau$</td>
<td>Shear Stress</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius</td>
<td>$\phi$</td>
<td>Scalar Quantity</td>
</tr>
<tr>
<td>$S$</td>
<td>Source Term</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Chapter 1 - Introduction

As the global population continues to follow an upward trend, it is estimated that the global population will reach 8.6 billion in 2030, 9.8 billion in 2050 and 11.2 billion by 2100 [1]. Food security is therefore a vital issue and is bound to become a far more widespread problem. The world supply of wild fish is continuously being depleted due to overfishing and poor management, such that a sustainable method of providing fish stock is needed. Aquaculture is projected to be the prime source of seafood by 2030 as it meets many environmental, economic and social sustainability goals [2]. Unfortunately, aquaculture has neither received the concentrated research effort that has been applied to many agricultural fields, nor has it enjoyed the same widespread use in comparison to traditional fishing industry [3]. Only since the mid-twentieth century has aquaculture started to transition from an art, in which it operated based on a common sense approach, into a dedicated science with a complex and multidisciplinary approach [4]. This recent transition in the operational methods has allowed larger industries to adopt more engineering and scientific methods to increase the yield while reducing waste and costs. This has left the small facilities and less developed countries to continue to operate based on old, inefficient, and ineffective methods of operation. There is a definite need for more investigations to improve the design and operational methods of aquaculture systems by utilizing engineering principles. Such investigations can lead to an increase in the sustainability and economics of these systems.

An airlift pump is a device that utilizes the principles of buoyancy of pressurized injected air at the base of a submerged riser tube to act as pneumatic pistons to lift the submerged water to the top of the riser tube [5]. These pumps have been used in the oil and gas industries, as well as in the waste water industry. The increase in knowledge and understanding of two-phase flow has made it possible to study and analyze the various parameters that affect the performance of an airlift pump. This has led to improved pump designs that maximize the efficiency and performance potential. The use of airlift pumps in aquaculture is also not entirely novel, as it has been reported that airlift pumps are the
second most common type of pumps used in aquaculture [6]. Research on the topic is still limited and a portion of it is somewhat outdated. There is a need to revisit this idea to further expand on the topic and provide an attempt at understanding the operation of airlift pump systems and their potential under operational conditions. Using airlift pumps in industries where a supply of pressurized air is readily available, such as in the aquaculture industry, can reduce the complexity of the system while enhancing its performance.

Computational fluid dynamics (CFD) is a very powerful design and flow prediction tool that can aid in studying and analyzing various fluid phenomena. This tool has been used continuously and adopted by a number of industries. Its ability to provide a good and accurate flow prediction has provided research and industry ways to study complex systems without the need of building physical models. This provides invaluable time and cost savings for many industries when testing and innovating designs and systems. Using CFD to study two-phase flow has been limited historically due to the difficulty and complexity of modelling multiple phases numerically, as well as the computational expense required to run such simulations. The current state of the art of the commercial and open source CFD solvers, as well as the advances in the computational power of modern central processing units, have been able to provide the capabilities to solve problems of that complexity and magnitude. This has resulted in the potential to expand on the work done in the field of two-phase flow. In the current study, a commercial CFD package (ANSYS Fluent 19.0) was used to study the two-phase flow occurring in an airlift pump system as well as to conduct free surface simulation of an aquaculture raceway.
1.1 Thesis Objectives

The objective of this work is to study the effectiveness of integrating an airlift pump in an aquaculture system during the design phase. This work includes modelling the performance of a dual-injection airlift pump and validating it with experimental results. A study of an aquaculture raceway was also performed to find the important parameters required for a successful operation. The knowledge gained from understanding each system allows for improvement in the design process of integrating the airlift pump into the raceway. Studying the effectiveness of the combined systems can provide an indication on potential designs for current and future aquaculture systems. A summary of this work's objectives are:

- Understand the hydrodynamic flow characteristics in a dual injection airlift pump.
- Study the two-phase flow phenomenon occurring in the airlift upriser pipe.
- Model the performance of an airlift pump system.
- Integrate an airlift pump system into an aquaculture raceway.
- Better design an airlift pump system for aquaculture systems.

1.2 Thesis Outline

This thesis consists of six chapters including the Introduction. Chapter 2 provides a comprehensive literature review on topics such as aquaculture, airlift pumps, two-phase flow modelling using computational fluid dynamics and the integration of airlift pumps in aquaculture. Chapter 3 discusses the experimental work done on this project. Chapter 4 covers the computational fluid dynamic modelling methodology used to represent the problem. Chapter 5 showcases the results and includes a discussion to provide further insights on the significance of the results. Finally, Chapter 6 provides concluding remarks as well as recommendations for future research.
Chapter 2 - Literature Review

This chapter will review the applicable literature that has been found on the topics of interest. Topics such as types of aquaculture systems, airlift pumps and the numerical work conducted in two-phase flow are covered to provide an overview on the work done for each topic. A brief introduction on aquaculture systems is presented to provide a background on their relevance to this work. A significant portion of this literature will focus on the theoretical, experimental and numerical work done in the field of two-phase flow and its application in the operation of airlift pumps. A brief portion will be dedicated to discussing the recent use of numerical simulations for aiding in the design of aquaculture systems. Finally, the gaps in the literature will be identified and the objectives of the study will be stated.

2.1 Aquaculture Systems and Engineering

Aquaculture is the farming and processing of fish, shellfish, and other aquatic animals combined with the raising of aquatic plants in fresh or salt water [7]. The practice of aquaculture is not a new phenomenon and in fact has been in existence for thousands of years. Ancient cultures in China, Japan, and other places in the Far East have practiced aquaculture for centuries [3]. The earliest thesis on aquaculture was published in 479 BC by Fan Li [8]. Ancient Greek and Roman authors also mentioned raising oysters and other aquaculture activities in their writing [3]. Today, aquaculture activities take place in various places such as offshore in the open ocean, land-based ponds, nearshore marines and indoor facilities all across the world [6]. There are many different aquaculture systems that are currently in use in industry. These systems range in their operational requirements and have different production capabilities that can be achieved. The three most common systems include: open-cage, flow-through and recirculating aquaculture systems.
2.1.1 Open-Cage Systems

An open-cage aquaculture system is a system that relies on setting up cages in open bodies of water both fresh, such as large lakes, or salty, such as seas and oceans. Open-cage aquaculture can be dated to as far back as the 1200s in some areas of Asia, and is currently a major form of aquaculture in Canada, Chile, Japan, and many other countries [9]. The United Nations Food and Agricultural Organization (FAO) reported that the total production from open-cage systems was approximately 3.4 metric tons in 2005 [10], with Canada claiming that almost 70% of its total production by volume was from cage systems [11]. The most common type of species grown in cages are high valued finfish such as salmon, seabass, snapper and rainbow trout [10]. The cages usually consist of an enclosure with a hinged lid at the top surrounded by a cage mesh, or of a net that is attached to a floating dock with an open top.
These systems are constrained by the amount of ecological support that the body of water can provide for the fish biomass. These limits occur since the oxygen supply is only provided through diffusion and photosynthesis by any local aquatic plants, and the fish waste products are only processed through the local bacteria culture operating at natural rates [14].

The other parameter that affects fish production is the amount of water movement the cages receive from waves or surface winds, as this action is what allows for the waste product to move away from the cages and for new clean and oxygenated water to be
brought into the cages [14]. These factors come into play when the design of an open-cage is discussed, as it determines the amount of fish stocking that can be sustained by the system.

The biggest advantage open-cage systems have over other aquaculture systems is that almost any body of water can be used as a setup location for a facility. The cost of construction is also relatively low compared to other systems, and the management of the cages is also simpler.

The major disadvantages of open-cage systems revolve around the biological handling of the fish mass in the system. Since most of these systems try to maximize the amount of fish they can produce, the high densities of fish stock result in issues such as rapid disease spread, poor localized water quality as well as fish abrasion where the fish stock gets bruised due to the friction between the fishes, which all may reduce growth [11]. Another disadvantage that is present in open-cage systems is the fouling of the cage netting. Fouling is the accumulation of unwanted material on a solid surface, which in this case is the aqua-cage [15]. Organisms such as algae, or sessile creatures such as sponges can grow on the surface of the cages and cause fouling. This leads to the need to have the cages scrubbed periodically [8]. Other disadvantages of open-cage systems include the possibility of fish escaping through the netting and also inviting predatory species to the cage sites [16].
2.1.2 Flow-Through Systems

Flow-through systems are considered semi-closed production systems, which in their simplest form are just shallow tanks that allow for water to enter from one end while it discharges through the other end. Thus, the water flows through the system while new water enters in a continuous cycle. Raceways are the most common flow-through systems and can be found in many different countries. Early flow-through systems consisted of placing various screens in different sections of a slightly sloped stream to confine the fish to that specific section [17]. Water would then naturally flow from one section to the other using gravity, thus providing the necessary water flow for the fish. These early systems had a lower water flow velocity to minimize and avoid erosion of the stream, which in turn lowered their fish carrying capacity. Modern flow-through systems are constructed out of concrete and have increased the amount of production by 25 to 40% using the same amount of water [17].
Figure 2-2- Flow-Through System Schematic (Top) and Operational Flow-Through Systems in the US (Bottom) [18]–[21]
Raceway systems operate on the principle that a constant flow of water is going through the system. This shortens the amount of time that the water is in the system, making it minutes rather than hours as compared to other systems. This constant flow of incoming water is the only source of oxygen added to the system. The water quality must also be good and the flow rate must be sufficient as it is needed to flush out all the waste that has accumulated in the system.

In terms of production, raceways are very intensive in terms of land and water use. On a per hectare basis, an excess of 300,000 kg of fish can be produced per year; however in order to produce 1 kg of trout, 98,000 liters of water is required [22]. Such requirements limit the type of production, as the potential geographical placement of flow-through systems are restricted to areas where a supply of flowing water is readily available and the fish species produced are sufficiently valuable and can be grown in a high density environment [17].

Raceways are most commonly located mountainous regions where gravity flows can be redirected to operate the raceway systems [6]. Other than the high production yield of these systems, other advantages include a greater ability to monitor the fish, easier harvesting, and forcing the fish to exercise which increases survival rates [3].

The biggest disadvantage of flow-through systems is the limited number of suitable geographical locations that meet all the requirements that are necessary for such systems. Flow-through systems are very effective at fish production, but due to the amount of water required to effectively run the system, very few places can benefit from such a system.

2.1.3 Recirculating Aquaculture Systems (RAS)

Another major system is the recirculating aquaculture system or RAS. These systems reuse the outlet water from the fish tank and re-introduce it to the system after it undergoes a filtration process. Theoretically this system does not require any new water to be added to the system, as ideally all the water is reused after the filtration process. In
practice this is impossible, as water is lost due to evaporation. As well, complete filtration of the water is unfeasible economically, so a certain amount of water is added during each cycle [9].

These systems are experiencing a very strong surge in popularity as they are seen as the best current solution to meet future fish demands in a sustainable matter. An example of such a boom is the Chilean industry which has grown from a US$159 million industry in 1991 into a US$2.3 billion one in 2007 using such systems [23]. These recirculating systems can be built in both indoor or outdoor facilities and therefore offer the greatest degree of flexibility amongst all the aquaculture systems. This has allowed these systems to spread for commercial, educational, research and even non-profit applications all over the world.
The most common RAS include recirculating raceways, circular tanks and D-ended raceways. These systems can vary in size and layout, but all recirculating systems must include at least one of each of the following components: a solid removal system, a biological filtration system and a pumping system that drives the whole process. A solid removal system is a very important first step in any filtration process. Solids, which may consist of organic or inorganic material, can accumulate in the fish tanks. These solids can affect the wellbeing of the fish as well as physically blocked pipes, pumps and filtration equipment [6].
A biological filter is a method of removing certain water substances using living organisms. The primary need for the filter is to convert ammonia to nitrite and nitrite to nitrate. Ammonia is a very toxic compound that is found in the metabolic waste of all fish, while nitrate is considered nontoxic to most aquatic species [3]. A pumping system finally drives the water from the biological filter and replenishes any water lost during the cycle. In practice, large recirculating systems have multiple stages of filtration and water purification to ensure that the water quality is high.

The benefits of a RAS include meeting the various sustainability, economic and engineering goals set by many world organizations. An indoor RAS is sustainable, infinitely expandable, environmentally compatible and has the ability to provide safe to eat, high quality fish throughout the year [23]. Recirculating systems also use 90 to 99% less water than other systems, and less than 1% of the land area, making them very environmentally friendly [23]. The biggest disadvantage facing a RAS is that the capital and operational costs are still too high to justify a major switch from other conventional systems. This applies to nearly all RAS facilities that grow salmon due to its high economic value [28]. Recirculating aquaculture systems are the responsible fish production method of the future, but better and more efficient designs are needed to make it an economically attractive option.
2.2 Airlift Pumps Under Two-Phase Conditions

Airlift pumps are special devices that lift fluids and mixtures of liquid and solid particles through partially submerged vertical pipes by utilizing only the buoyant forces of the air being injected into the pipe. Airlift pumps are used in various fields ranging from the petrochemical and oil industries to the aquaculture and food processing industries. The first airlift pump was invented by Carl Loscher, a German engineer, in 1797 [29]. These systems were initially used for deep well pumping and mining operations. It was not until the mid-19th and early 20th centuries that they started being used in sewage treatment plants. Airlift pumps were first introduced to the U.S. in a Pennsylvania oil field in 1846 [30].

All airlift pump systems consist of three parts. The part below the air injector is called the suction pipe. Next, there is the air injector. Everything above the air injector and up to the discharge point is called the upriser or riser pipe. There are three main types of airlift pump systems, each with its own unique upriser pipe geometry. They are the ordinary type, the step type, and the tapered type.
The ordinary type is the simplest and most common, in which the upriser pipe has a constant cross-sectional area throughout its length. The step type airlift system first introduced by Kumar et al. [31], has a multi-step upriser pipe, in which each section has a different cross sectional area than the other sections and is usually put in order of increasing diameters. Kumar et al. as well as Karimi et al. [32] found that this system improved the performance of the airlift system. Karimi also proposed an ideal height and diameter ratio for the stepped system to better improve the performance of the system. Finally, the tapered system, which was also introduced by Kumar, has an upriser pipe that slowly diverges creating a larger cross-sectional area along its length. Hanafizadeh et al. [33] as well as Zaraki et al. [34] further investigated this system and found that the system’s efficiency increased with an increase in the tapering angle until it reached three degrees, at which point the output of the system started to decrease.

Despite their lower efficiency when compared to mechanical pumps for some applications, these systems have many practical advantages, such as a lower initial cost,
lower maintenance, lower operation costs, easy installation, portability, freedom from clogging, small space requirements, simplicity of design and ease of construction [30].

2.2.1 Theory of Operation

The operation of an airlift pump involves injecting a compressed gas at a certain depth below the fluid level in a partially submerged pipe. This creates an interaction between the gas and the liquid, thus creating a two-phase flow system. Since most gases are far less dense than liquids, the gas starts to rise in the upriser pipe. The forces acting on the gas-liquid mixture are buoyancy, which acts as the lifting force, as well as inertia and gravity, which act as the opposing forces. When the lifting forces are large enough, the liquid rises along the pipe to the level at which the acting forces are equal.

The performance of an airlift pump is tied to the submergence level of the airlift system, the upriser pipe diameter and the two-phase flow pattern experienced in the pipe. The submergence level is typically quantified through a parameter called the submergence ratio. This ratio represents the amount of static lift in comparison to the total lift of the airlift system. The submergence ratio has been noted in the literature to be the most significant parameter affecting the operation of airlift pump systems. This was studied experimentally, and it was found that at higher submergence levels, the performance and efficiency of the pump increased significantly [35],[36]. The airlift pump efficiency is defined by the amount of work needed to raise the liquid up the riser pipe, divided by the isentropic expansion work by the gas as seen in equation (1) [37].

\[
\eta = \frac{\rho g Q_l (L - H_s)}{P_a Q_g \ln \left( \frac{P_1}{P_a} \right)}
\] (1)
The diameter of the upriser pipe is also a parameter that affects the performance of an airlift pump system, as piping size can affect the forces that act on the bubbles. In small pipes with a diameter below 20 mm, the effects of the surface tension forces on the bubbles becomes more profound, as discussed by Nickens & Yannitel [38] and Zukoski [39]. Reinemann [5] also found that besides the forces described by Nicklin [37], there are forces generated by the vortices created in the wake of the rising bubbles. These forces also effect the efficiency of the airlift pump system. In larger pipes, these forces are small and are considered negligible, as determined by Richardson and Higson [40],

**Figure 2-5-Airlift Pump System Schematic**

The diagram shows the schematic of an airlift pump system, with the upriser pipe marked by the symbol $H_s$ and the discharge outlet at $L$. The air supply is connected to the system at the top right, indicating the flow direction and pressure distribution within the system.
Husain [41], Jeelani [42] and others. Kim et al. [43], Awari et al. [44], Hanfizedeh [33], and Fan et al. [45] all found that an airlift pump’s efficiency increased with an increase in the upriser pipe’s diameter due to the reduction of the forces acting on the bubbles.

Depending on the input flow conditions, different types of two-phase flow regimes can develop inside the upriser pipe. Most of the literature on gas-liquid flows has divided two-phase flow regimes in a vertical pipe into four major flow patterns: bubbly, slug, churn, and annular flow patterns [46]. The flow pattern in the upriser pipe is an important factor in determining the efficiency point of an airlift system. The different types of airlift systems were designed to preserve a favorable flow pattern in the pipe over a larger range of operating conditions, while limiting the unfavorable ones.

Figure 2-6- Two-Phase Flow Patterns
Bubbly flow is characterized by the distribution of discrete bubbles in a continuous liquid phase. The bubble sizes can range from small and spherical, to larger ones which are still smaller than the pipe’s diameter and have a spherical cap and flat tail [47]. The different shapes and trajectories that the bubbles take are due to interactions in the surface tension, viscosity, inertia and buoyant forces [48]. This flow pattern is desirable for mass transfer [49]–[51], high-pressure evaporators [52]–[54] and many other applications.

In an airlift system, it has been observed that bubbly flow is not ideal for general pumping purposes [29], [55]. The lift forces acting on the bubbles in this pattern are smaller than those pulling down; therefore, it is only effective for pumping liquids in cases of high submergence ratios. Due to this, research on airlift pump performance has mostly neglected bubbly flow. Clark et al. [56] presented a theoretical model for the operation of airlift pumps in bubbly flow, but it was not accompanied by any experimental validation data.

Slug flow is characterized by a series of individual large bubbles of varying lengths with diameters near that of the pipe. [47]. Each air bubble is separated by a liquid flow which is called liquid slug. In airlift pump systems, slug flow is the most desirable and most encountered flow pattern. The air bubbles act as pneumatic pistons that push the liquid slugs in front, while at the same time creating suction behind that draws in more liquid [29]. The literature is rich with experimental and theoretical work done on airlift systems operating in slug flow in both large diameter pipes [37], [40]–[42], [57]–[60] and small diameter pipes [38], [39], [61]. Hanafizadeh et al. conducted a flow visualization experiment regarding the flow patterns occurring in an airlift pump, and found that slug and slug-churn flows are the most efficient flow patterns for operation [62].

Churn flow occurs when the air flow rate is increased and effectively starts growing and breaking down the slugs, creating a chaotic and unstable flow. In churn flow, the lifting forces of the bubbles are still larger than the pulling forces, and so it is also a desirable
flow pattern for the operation of an airlift pump. Kassab et al. found experimentally that when the flow pattern is in transition between slug and churn (slug-churn) flow, that the maximum water flow rate was achieved [36]. De Cachard [63] proposed a slug-churn model to predict the performance of small diameter airlift pumps (up to 40 mm). Since the most dominant flow pattern in small airlift pumps is slug and churn flow [64], this model took into account the frictional and surface tension forces that have a much more adverse effect in small diameter pipes.

If the air flow rate increases beyond that of churn flow, annular flow results. An annular flow pattern is characterized by a continuous liquid film flow along the wall of a pipe while the gas flows in the central core [48]. This flow is not desirable in airlift pump systems since the pump’s efficiency decreases rapidly. The model developed by Stenning and Martin [60] shows that in annular flow, the liquid flow rate should decrease, while Kassab et al. [36] and Hanafizadeh et al. [62] found experimentally that the liquid flow rate remains stable.

The air injection method was also identified as a parameter that can improve the performance and efficiency of an airlift pump, as well as maintain favorable flow patterns in the upriser pipe. Fan et al. [45] mentioned the need to investigate the optimal design for an air injector as this has a considerable effect on upwelling efficiency in ocean waters.

Khalil et al. [65] tested nine different air injection foot piece designs at four different submergence ratios to determine the performance and efficiency of an airlift pump. It was found that certain designs improved the efficiency of the airlift pump by as much as 21% at the same submergence ratio. Depending on the submergence ratio, a suitable design and hole configuration can also be found to improve the performance [65].

Hu et al. [66] also studied the effect of the air injection method in an airlift pump operating under the three-phase flow conditions of sand-air-water. Hu found that with a higher number of air injectors, the volume flow rate of the water and the solids increased as well [66].
A new injection method was developed by Ahmed and Badr [67]. In this method, the airlift combined two different air injection methods into one pump. The efficiency of this new design was improved by 30% to 90% depending on the submergence ratio. Ahmed et al. [68] further investigated various air injection methods and found that pulsating the air injection at a certain frequency further improved the performance of the airlift pump.

### 2.2.2 The Use of Airlift Pumps in Aquaculture Systems

The potential benefits of using airlift pumps in the aquaculture industry have been recognized since the seventies. Spotte [69] listed the benefits of using an airlift pump in aquaculture. These included a lower initial cost, lower maintenance, easy installation, portability, and freedom from clogging. Wheaton [3] recognized the airlift pump’s ability to aerate and oxygenate the water as it simultaneously pumps it. Although all aquaculture facilities are required to aerate water to provide a sufficient oxygen supply to the fish, most facilities use separate systems to first pump and then aerate their water. An airlift pump can combine both functions while being less costly and more economical in the operational period of the facility [3]. This has led to a burst of research into the performance characteristics of airlift pumps in an aquaculture setting.

Parker and Suttle [70] were one of the earliest researcher groups to specifically design an airlift pump system for the purpose of aquaculture. They conducted experiments on airlift pumps with diameters ranging from 3.75 to 30 cm while having the pump at 100% submergence. Wurts et al. [71] also conducted experiments using airlift pumps made from PVC piping with diameters ranging from 7.6 to 15.2 cm. The experiments found that the airlift pump’s ability to aerate and agitate solid particles to be very suitable for floating raceways, open cages and RAS.

Barrut et al. [72] expanded the study on RAS to include the effects of pumping fresh and salty water using airlift pumps. It was found that airlift pumps performed better in fresh water due to a difference in water quality that effected bubble coalescence. It was also found that in low water lift cases, i.e. below 0.3 meters, the airlift could save up to 50% of
the energy required by a centrifugal pump [72]. Loyless and Malone [73] conducted a study on an airlift pump's oxygen transfer rate as well as its carbon dioxide stripping ability. It was found that both the standard oxygen transfer rate (SOTR) and the carbon dioxide stripping rate increased with an increase in the air injected into the pump.

Barrut et al. [74] expanded on the research by studying mass transfer efficiency using salty as well as fresh water and found that the salinity of the water did not affect the efficiency. Barrut [74] reported that the bubble size and air flow had a greater effect on the mass transfer coefficient. Airlift pumps can also be used to pump fish from one location to another [9]. With its virtue of having no moving parts, airlift pumps can provide a safe way to transport fish stock from one location to another without causing any damage to the fish, unlike that which can be found using other types of pumps [9].

![Airlift Pump for Transporting Fish](image-url)

**Figure 2-7** - Airlift Pump for Transporting Fish
2.3 Two-Phase Flow Numerical Modelling Using CFD

The use of computational fluid dynamics (CFD) as a design and a prediction tool has been increasing in popularity. This is mainly due to the growing computational power available in modern processors. Regardless of computational power, modelling of two-phase flow is still complex due to the nature of multi-phase flow. The amount of work done using CFD is therefore much smaller compared to the experimental or pure numerical work available. Most of the available research has focused on the operation and design of bubble columns, due to their high economic potential but difficult scaling process. Work on two-phase flow in vertical and horizontal pipes is also present but is far less frequent. Work done specifically on airlift pump operation and performance modelling is very scarce and limited. This small pool of research is attributed to the need to develop specific multi-phase CFD code to better model the flow patterns, as well as the coalescence and breakup, of bubbles. While commercial CFD codes now offer more powerful multi-phase models, they do not yet provide the same control over parameters in comparison to in-house CFD codes.

2.3.1 Numerical Modelling of Bubble Columns and Airlift Reactors

Bubble columns are a type of reactor that is used in a wide range of chemical processes ranging from the manufacturing of fine chemicals, to coal liquefaction, and the production of single cell proteins [75]. A sparger at the bottom of the column pipe introduces gases into the liquid in the bubble column. Airlift reactors are an alternative to mechanically agitated reactors and are used to facilitate water circulation and mass transfer in bio reactors by using an airlift pump [76]. The interest in this topic of research is that bubble columns and airlift reactors share many similarities to airlift pumps, as the sparger can be seen as an equivalent to the air injector, while the liquid column is equivalent to the upriser pipe in the airlift.
There are various modelling approaches found in the literature. There are one-dimensional models by Rice and Nicholas [77], Rice and Burns [78] and Vitankar and Joshi [79]. These works attempted to predict the gas hold up, the liquid and gas velocities as well as the pressure drop in a bubble column. A two-dimensional model developed by Webb et al. [80] was created by coupling a buoyancy induced liquid flow with a turbulence induced gas dispersion using the Euler-Lagrange method to study the flow pattern of the liquid and the bubbles inside a rectangular bubble column.

Sokolichin and Eigenberger [81], [82] and Borchers [83] also developed a model using an Euler-Lagrange as well as an Euler-Euler approach. These studies tested a laminar and a K-ε turbulence model to study the flow structure and the liquid velocity inside the bubble column. They found that the Euler-Lagrange approach presented a more physical interpretation of the experimental work but was very computationally intensive, while the
Euler-Euler approach was much less computationally expensive and provided a good agreement with the experimental data. It was also found that the laminar solution was directly tied with the mesh quality, where the higher number of elements used created a better solution. In contrast, the K-ε turbulence model overestimated the effective viscosity and dampened out the dynamic characteristics of the flow.

Sanyal et al. [84] also modelled a 2D bubble column with an axisymmetric boundary using a commercial CFD package, which combined a Eulerian approach with an Algebraic Slip Mixture Model (ASMM). It was found that the model provided reliable approximations of the time-averaged flow as well as the gas patterns experienced.

Van Baten and Krishna [85] conducted a 2D simulation with an axisymmetric boundary of a bubble column operating in both the homogeneous (bubbly flow pattern) and heterogeneous (slug, churn flow pattern) flow regimes. The simulation was done using a commercial CFD package and used the Eulerian-Eulerian two-fluid model, with a K-ε turbulence closure. The results showed that in the heterogeneous flow regime, the large bubbles tended to concentrate near the central core of the column, while in homogenous flow the bubbles were more distributed over the column. This result is consistent with what has been found in the literature regarding the operation of airlift pumps, as the larger bubbles (slug, churn) have much stronger lifting forces acting on them. These bubbles would therefore move directly in an upward vertical fashion.

The addition of a third-dimension into a bubble column model further complicates the simulation, as it also becomes much more computationally intensive. Ekambara et al. [86] studied a bubble column using a 1D, 2D and 3D approach to find which approach was most suitable to model the operation of the bubble column. Ekambara used models developed by Vitankar et al. [79] for the 1D simulations, and Ekambara and Joshi for the 2D [87] and 3D [88] respectively, along with a K-ε turbulence closure model. These models were compared with the experimental work of various researchers. It was found that the 1D, 2D and 3D models had a good agreement with the experimental data for the
liquid velocity and gas holdup profiles. Only the 3D model, however, was able to predict the eddy diffusivity with a good agreement.

Mudde and Van Den Akker [87] similarly studied the differences between a 2D and 3D model of an internal airlift loop. The two-fluid model that was used was derived through conditionally ensemble averaging the local conservation equations of single-phase flow as found in Delhaye [88] and Drew [89], with a K-ε turbulence closure. This formulation presents the mass, momentum and energy balance results in the form of averaged quantities. It was observed that the 2D model over predicted the amount of circulation occurring in the reactor, which affected the gas holdup profiles that were found. This effect was apparent due to the lower friction present in the 2D model. The 3D model was better suited for predicting the circulation and the gas holdup, as well as the superficial velocity profiles in the riser pipe.

Work done by Pfleger et al. [90], Sokolichin et al. [82], and Delnoij et al. [91] all agree that 3D models are far more suitable for simulating and capturing the operations of a bubble column. Pfleger and Becker [92] conducted a 3D simulation of a bubble column operating in the homogeneous flow regime, using a commercial CFD package. The two-fluid model used was the Eulerian-Eulerian model, with a K-ε turbulence enclosure. The timed averaged data recorded for the liquid velocity and gas holdup profiles showed good predictions when compared with the experimental data. Since the operation was in the homogenous regime, the dominant flow pattern observed was bubbly flow. The model's capability of capturing transitions in the flow patterns was therefore not found.

Akhtar et al. [93] simulated a 3D bubble column to study the behavior of a continuous chain of bubbles under different inlet conditions. Three different diameters of air inlets were experimentally tested and numerically modeled to see the resulting effect in each case. The simulation utilized the Volume of Fluid (VOF) two-phase model, with K-ε turbulence closures. The simulation showed that the bubble rise velocity and the bubble size increased with an increase in the superficial velocity. An increase in the bubble size
was also observed with an increase in the size of the air inlet. The bubble rise velocity, shape, typical rise trajectory and holdup that was obtained from the simulation had a very close agreement with the experimental results.

Suhaimi and Nasir [94] also used the VOF model to study the effect of flow pattern on biodiesel production in a 3D bubble column. The simulation used triglyceride as the liquid and MeOH vapor as the gas in the system. The results of the velocity and gas holdup profiles showed that the simulation results over predicted by nearly 7% when compared to the experimental data. Tabib et al. [95] as well as Ekambara and Dhotre [96], presented work analyzing the effects of different turbulence closure models that are available in commercial CFD packages on the results of a 3D bubble column.

Tabib et al. [95] simulated a bubble column found in the experimental work of Bhole et al. [97] using the Eulerian-Eulerian two-fluid approach with the K-ε, RNG, and LES turbulence models. The results showed that near the air inlet, none of the models were able to predict the axial velocity or gas holdup profiles, but as the axial distance from the inlet grew, the prediction’s capabilities improved. The RSM model did not show any improvement over the K-ε model in predicting the velocity profiles. The LES model succeeded in capturing the average behavior of the flow, but over predicted the kinetic energy and stress profiles in some cases. It was concluded that the LES model was much better than the K-ε or RNG at capturing the instantaneous phenomena, but that the additional computational power needed to use LES or RNG will not add any significant value to the information obtained by the K-ε model for averaged flows in a 3D bubble column.

Ekambara and Dhotre [96] further expanded on this topic by adding the K-ε RNG as well as the K-ω model to the aforementioned turbulence models. The findings of Ekambara [96] agree with Tabib et al. [95], as LES provided the closest prediction of the results to the experimental data. It was also noted that the K-ε model was better at predicting the average axial velocity and gas holdup profile than the K-ω model, but that the K-ω model
was better at predicting the turbulent kinetic energy. Ekambara and Dhotre [96] concluded that for an average flow simulation, models such as the K-ε and K-ω are preferable due to their lower computational intensity.
<table>
<thead>
<tr>
<th>Author</th>
<th>Case</th>
<th>Dimension</th>
<th>Multi-Phase Model</th>
<th>Turbulence Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rice and Nicholas 1990 [77]</td>
<td>Predicting flow circulation</td>
<td>1D</td>
<td>-</td>
<td>Prandtl Model</td>
</tr>
<tr>
<td>Rice and Burns 1997 [78]</td>
<td>Predicting gas holdup and liquid velocity</td>
<td>1D</td>
<td>-</td>
<td>K-ε</td>
</tr>
<tr>
<td>Vitankar and Joshi 2002 [79]</td>
<td>Predicting gas holdup, fluid and gas velocity, pressure drop</td>
<td>1D</td>
<td>-</td>
<td>Sato et al.</td>
</tr>
<tr>
<td>Webb et al. 1992 [80]</td>
<td>- Rectangular bubble column: 0.3m wide</td>
<td>2D</td>
<td>-</td>
<td>K-ε</td>
</tr>
<tr>
<td></td>
<td>- 7 air orifice inlets: 2E-6 m^3/s</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sokolichin and Eigenberger 1994, 1999 [81], [82]</td>
<td>- Rectangular bubble column: 0.5m wide</td>
<td>2D</td>
<td>-</td>
<td>Laminar, K-ε</td>
</tr>
<tr>
<td>Sanyal et al. 1999 [84]</td>
<td>- Cylindrical bubble column: 0.19m diameter</td>
<td>2D</td>
<td>Euler-ASMM</td>
<td>K-ε</td>
</tr>
<tr>
<td></td>
<td>- Velocity inlet: 0.02 to 0.12m/s</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Van Batten and Krishna 2002 [85]</td>
<td>- Rectangular bubble column: 0.05 m wide</td>
<td>2D</td>
<td>Eulerian-Eulerian</td>
<td>K-ε</td>
</tr>
<tr>
<td></td>
<td>- Velocity inlet: 0.006 to 0.08 m/s</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Author</td>
<td>Case</td>
<td>Dimension</td>
<td>Multi-Phase Model</td>
<td>Turbulence Model</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>----------------------------------------------------------------------</td>
<td>-----------</td>
<td>-----------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>Mudde and Van Den Akker, 2001</td>
<td>- Airlift loop reactor: 1.02m height, 0.1m width, 0.12m depth</td>
<td>2D, 3D</td>
<td>Two-Fluid Ensemble Averaging</td>
<td>K-ε</td>
</tr>
<tr>
<td>Pfleger and Becker, 2001</td>
<td>- Cylindrical bubble column: 0.288m diameter 2.6m height - Velocity inlet 0.0015 to 0.2m/s</td>
<td>3D</td>
<td>Eulerian-Eulerian</td>
<td>K-ε</td>
</tr>
<tr>
<td>Akhtar et al., 2007 [90]</td>
<td>- Cylindrical bubble column: 0.1m diameter - Multi diameter velocity inlet: 0.02 to 0.1m diameter 1E-4 to 0.1 m/s</td>
<td>3D</td>
<td>VOF</td>
<td>K-ε</td>
</tr>
<tr>
<td>Tabib et al., 2008 [95]</td>
<td>- Cylindrical bubble column: 0.15 m diameter - Velocity inlet: 0.012 to 0.096 m/s</td>
<td>3D</td>
<td>Eulerian-Eulerian</td>
<td>K-ε RSM LES</td>
</tr>
<tr>
<td>Ekambara and Dhotre, 2010</td>
<td>- Cylindrical bubble column: 0.15m diameter - 25 air velocity inlets: 0.02 m/s</td>
<td>3D</td>
<td>Eulerian-Eulerian</td>
<td>K-ε RNG K-ω RSM LES</td>
</tr>
</tbody>
</table>
2.3.2 Numerical Modelling of Two-Phase Flow in Vertical Pipes

While there is a plethora of work available on two-phase simulations related to bubble columns, there has been much less work done specifically for the purpose of studying two-phase flow in detail using CFD. Studying this type of work will give a much better picture of the multi-phase models that are in use. These works involved studying the flow pattern and other flow parameters in more detail than is provided in the previous studies.

Mao and Duckler [98], [99] developed a 2D numerical model to study the velocity field around a single slug bubble in a vertical pipe. This model was used to predict the rise velocity of the bubble. The model considered the inertial and viscous terms as well as the effect of gravity. The model assumed the flow to be laminar [98], but was later adjusted by adding a K-ε turbulence closure model [99]. Mao and Duckler’s model started by stating the geometry of the domain as well as the slug bubble’s initial velocity.

The model would then predict the velocity and the pressure profiles around the slug, as well as the shape of the slug as it travelled upwards. The model indicated that theoretically there are multiple solutions that exist for each case, as multiple velocity fields and bubble shapes satisfy the Navier-Stokes equation. A criterion was used to set the rate of change in curvature to zero, and it was found that only one rise velocity satisfied this criterion. The results from this case were found to have an excellent agreement with the experimental data collected.

Taha and Cui [100] conducted a similar simulation of a slug bubble in a 2D domain using a commercial CFD package. The simulation studied the hydrodynamics of a single passing slug bubble in both a vertical tube and in a tube on a 20-degree tilt from the vertical axis. The simulation showed the changing shape of the slug as it moved along the pipe. The flow around the slug was also observed, as well as the wake region that was created underneath the slug.

This case truly showed the capability of the VOF model to model the interactions occurring between the two-phases in a vertical pipe. Abdulkadir et al. [101] conducted a study to
compare the experimental and CFD void fraction results of slug flow in a vertical pipe. Experimentally, the void fraction was collected along the height of the pipe using electrical capacitance tomography. A 3D model of the experimental setup was imported into a commercial CFD package. While the void fraction data showed a good agreement with the experiments, the liquid film thickness developed alongside the slug bubbles was shown to be under predicted by at least 15%.

Da Riva and Del Col [102] conducted a simulation to study churn flow in an air-water and R134a refrigerant vapor-liquid system. The study attempted to replicate the ring-type waves typical in churn flow, by focusing on the effects of a porous media inlet radially injecting air into the vertical pipe. The simulation results showed that the liquid motion had a pulsating nature due to a higher flow reversal than that found experimentally. Qualitatively, the overall flow pattern for both the air-water and the R134a cases matched what was found experimentally.

Parsi et al. [103] also attempted to numerically model churn flow in a vertical pipe. A hybrid model developed by Yan and Che [104] was used. This hybrid model combined the robustness of the Eulerian-Eulerian model, with the interfacial tracking capabilities of VOF. The hybrid model reduced the numerical diffusion as well as removed any closure correlation required by the Eulerian-Eulerian model [104]. Parsi et al. [103] captured the void fraction profile experimentally using a wire mesh sensor and compared that data to the numerical simulation. The simulation results showed a good agreement with the experimental data, but Parsi indicated that this might be misleading as churn flow can have different local flow structures for the same void fraction values.

The hybrid model was also able to capture the physical flow structures but not in all flow cases. When the length scale of the bubbles was large, the model was able to predict such flow structures, but when the superficial gas velocity increased and the bubble length scale became smaller, the model had more difficulty representing the flow structure. Two different turbulence models were tested (K-ε RNG and K-ω SST) to find
the effects of each on the results. The K-ω SST model showed that it was able to predict the flow structure better than the K-ε RNG model, but the average void fraction data was similar.

Parvareh et al. [105] conducted a numerical simulation to see whether a commercial CFD package would be able to capture the changing flow patterns in a vertical and horizontal pipe. A comparison of the void fraction data was done between the numerical solutions obtained and the experimental data recorded. The void fraction data was collected experimentally using electrical resistance tomography at a single location in the pipe, and the flow structure was captured using a digital camera. In the vertical setup, the plug (bubbly-slug transition), slug and annular flow patterns were selected to be of interest. The CFD results were compared with the digital images captured and reconstructed tomograms from the experimental setup. It was observed that each of the flow patterns were distinguished and had very good qualitative agreement with the experimental images.

An important work conducted by Wahba et al. [106] studied the performance of an airlift pump using CFD. The work included the modelling of a 3D airlift pump with a riser pipe, operating at two different submergence ratios and various mass airflow injections to complete a performance curve. Volume of Fluid and Large Eddy Simulation were chosen as the multi-phase and turbulence models respectively. The operating conditions used in the simulation were obtained from experimental work done by Kassab et al. [36]. The results were later compared to that study as well as with the analytical models developed by Reinemann [107] and Stenning and Martin [60]. The operation of an airlift pump was mainly in the churn and churn-annular region. Wahba et al. [106] reported that using VOF and LES resulted in the simulation accurately predicting the flow patterns, as well as the transition regions when the air flow rate increased. It was suggested that contributions to improve the LES model, such as including the sub filtered portion of the surface tension forces, would further improve the
simulation results. The literature has showed that there are two major multi-phase models that are used: VOF and Eulerian-Eulerian.

The discussion of which model to use has led Guerrero et al. [108] to conduct a study of a 3D vertical pipe using both multi-phase models and to compare the results of each. The tests included operating in bubbly, slug-churn and annular flow patterns. The results showed that in bubbly and slug-churn, the Eulerian-Eulerian model was more accurate than the VOF by 5%, while in annular flow both models struggled equally to provide accurate predictions of the flow parameters. It was also noted that only the VOF model was able to distinguish the discontinuity in the air-water flows, thus making it the only suitable model for flow pattern identification.
<table>
<thead>
<tr>
<th>Author</th>
<th>Case</th>
<th>Dimension</th>
<th>Multi-Phase Model</th>
<th>Turbulence Model</th>
</tr>
</thead>
</table>
| Mao and Duckler 1990, 1991 [98], [99] | - Vertical tube: 0.05 m diameter  
- Single slug bubble: 0.17 to 0.242 m/s | 2D        | -                 | Laminar K-ε      |
| Taha and Cui 2005 [100]  | - Vertical tube: 0.019 m diameter  
- Single slug bubble: 0.81 to 1.63 m/s | 2D        | VOF               | K-ε RNG          |
| Zheng et al. 2007 [109] | - Vertical tube: 0.036 m diameter  
- 0.5 m height  
- Effects of a slug bubble on the mass transfer coefficient | 2D        | VOF               | K-ε              |
| Da Riva and Del Col 2009 [102] | - Vertical tube: 0.01 to 0.032 m diameter  
- Simulation of churn flow for an air-water and R134a refrigerant vapor-liquid system | 2D        | VOF               | K-ε RNG          |
| Ratkovich et al. 2013 [110] | - Vertical tube: 5.2E-4 m width  
- 9.5E-3 m height  
- Slug flow in Newtonian & non-Newtonian fluids | 2D        | VOF               | Realizable K-ε   |
<table>
<thead>
<tr>
<th>Author</th>
<th>Case</th>
<th>Dimension</th>
<th>Multi-Phase Model</th>
<th>Turbulence Model</th>
</tr>
</thead>
</table>
| Parvareh *et al.* 2010 [105] | - Vertical and horizontal tubes  
- Single air inlet:  
0.08 to 23.7 m/s | 3D | VOF | Laminar |
| Wahba *et al.* 2014 [106] | - Airlift pump:  
56 injectors: 3 mm diameter  
- Upriser pipe:  
3.75 m length  
25.4 mm diameter  
- Submergence ratios: 48.4% and 74% | 3D | VOF | LES |
| Abdulkadir *et al.* 2015 [101] | - Vertical tube:  
0.067 m diameter  
6 m height  
- Study the void fraction of slug flow along length of pipe | 3D | VOF | K-ε |
| Parsi *et al.* 2016 [103] | - Vertical tube:  
0.076 m diameter  
- Velocity inlet:  
10.3 to 33.9 m/s superficial gas velocity  
0.3 to 0.79 m/s superficial liquid velocity  
- Churn flow with a transition to annular flow | 3D | Hybrid Eulerian-Eulerian | K-ε RNG K-ω SST |
| Guerrero *et al.* 2017 [108] | - Vertical tube:  
0.05 m diameter  
- Compare the performance of the VOF and Eulerian-Eulerian models | 3D | VOF Eulerian-Eulerian | K-ε |
2.3.3 Numerical Modelling of Free Surface Flow

Free surface flow refers to the dynamic interaction between air and water at a surface that is subject to zero parallel shear stress [111]. This type of flow is also considered two-phase flow and it usually occurs in an open and often large flow domain. Flow phenomena such as waves, sloshing, spilling or mixing are all examples of free surface flows. The simulation of free surface flows has vast engineering and non-engineering applications ranging from the environmental such as simulations of open channel flow [112]–[116], civil such as the simulation of water in spillways and dams [117]–[120], marine applications [121]–[124], mechanical such as the sloshing of petroleum in process vessels and transportation tanks [125]–[129], and many others. This makes covering the topic of free surface simulations a slightly challenging task, as each of these applications share many similarities while also predicting very different parameters.

No work was found on free surface flow simulation in aquaculture facilities such as raceways and tanks. The simulation case presented in this work is of a raceway operating using an airlift pump system. This case resembles the work on spillways and open channel flow. This section will present some of the relatable work on free surface flow in the literature.

Rhee et al. [130] conducted a numerical simulation of the free surface waves around a surface-piercing foil. The study analyzed the wave generation of a submerged foil in an open tank using the VOF model. The study of the wave motion as well as the spilling and breaking of the waves was observed, and the forces acting on the foil were acquired. The study reported that the VOF method performed well for a wide range of free surface wave types [130]. Chanel and Doering [118] as well as Andersson et al. [120] both used the VOF model to simulate the flow field of various spillways.

Chanel and Doering [118] assessed the modelling capabilities of the VOF by simulating three spillways of different sizes that existed in the field. The headwater level and discharge rates were the parameters of interest in this simulation. It was found that most
of the simulation data matched the experimental data recorded in the field. The errors were mostly found in cases of small experimental discharge, where the numerical model underpredicted the values.

Andersson et al. [120] on the other hand conducted experimental and numerical work on a scaled down model of a spillway. The purpose was to study the water surface level and the velocity at the spillway gates under two different conditions: with one gate open and with all the gates open. The experimental data collection was done using a velocimeter, while the numerical model was setup using VOF and the SSG and K-ε turbulence models. The results reported for the water surface level showed good qualitative agreement with the experimental measurements. The velocity field and measurements also showed good agreement with the experimental data. The two turbulence models showed very similar results, but the SSG was able to represent flow separation better.

Cheng et al. [131] proposed using the mixture model along with the RNG K-ε turbulence closure model to study two-phase flow over a stepped spillway. The purpose of the work was to predict the velocity, pressure and flow structure in the spillways to help improve the design process. The results showed that the mixture model was able to capture the entrained bubbles and void recirculation in the flow as it passed through each step. Cheng also reported that the RNG K-ε turbulence model was better suited for capturing the recirculating flows occurring over the steps.

2.4 The Use of CFD in Designing Aquaculture Systems

Aquaculture engineering has also benefited from the use of computational fluid dynamics. The design process of some aquaculture facilities has been enhanced by implementing CFD studies. This has improved the effectiveness of these facilities when built and operated. Throughout the literature, it was apparent that the most studied effect revolves around improving the settling effectiveness of solid particles, such as fish excrement or unused feed pellets. There was no work found that used two-phase simulations to study
the flow structure in aquaculture systems; the work available was limited to single phase simulations only.

Huggins et al. [132] simulated various designs of an aquaculture raceway to find which design improved the settling effectiveness of solid particles. This was important as the designs involved the addition of a screen across the raceway which disturbed the flow in the raceway. The results achieved increased the settling effectiveness by 10% while also maintaining good flow velocity through the system.

Labatut et al. [133] similarly studied the residence and settling time of various sized particles in a mixed-cell raceway. The study focused on the velocity profiles in the raceway, as they are the biggest factor on the residence time that a particle experience. This work helped in identifying the velocity profiles required to remove any solid particles within a certain size range.

Yao et al. [134] also used CFD to study the hydrodynamics of an octagonal tank used in a recirculating aquaculture system. The study was similar to Labatut et al. [133] as the main focus of the work was on the velocity profiles experienced in the tank, and its effect on the movement of different sized particles. Liffman et al. [135] conducted a numerical simulation of a circulating raceway to find a design that would reduce the amount of energy loss experienced at the bends in conventional circulating raceways. The new design proposed was able to reduce 87% of the energy loss experienced, while also improving the mixing capabilities by reducing the stagnation regions. Liffman [135] reported that implementing such a design could allow for larger raceway construction while lowering the relative capital costs by nearly 5%.
2.5 Gaps in the Literature

As can be seen from the literature provided, there has been much work done on studying aquaculture systems, airlift pumps and the integration of the two systems. Unfortunately, all the work presented has studied each topic independently rather than combining all the topics. The one work that attempted such an integration only focused on the experimental part and no numerical work was done. The current study attempts to fill that gap by introducing computational simulations to improve the design process of aquaculture systems using airlift pumps while at the same time modelling the performance of the pump through CFD.

2.6 Study Objectives

The objective of this study is to present a better design method for an airlift pump system to enhance its performance in an aquaculture raceway. This is done by better understanding each component of the system of interest. Studying a dual injection airlift pump illustrates the flow dynamics in the injectors and their respective effects on the flow domain.

The study of the entire airlift pump system provides a new method of modelling the performance of an airlift pump operating under two-phase flow conditions. Finally, integrating all the components of the work into an aquaculture raceway provides a test case to study the effectiveness of adding airlift pumping systems to traditional aquaculture systems. Figure 2-10 provides a schematic of the complete system as well as the individual blocks that will be analyzed to complete the system analysis. The end result would provide a method of improving the design of aquaculture facilities using airlift pump systems.
Summary of Objectives:

- Understand the hydrodynamic flow characteristics in a dual injection airlift pump.
- Model the performance of an airlift pump system.
- Integrate an airlift pump system into an aquaculture raceway.
- Better design an airlift pump system for aquaculture systems.

Figure 2-9: Schematic of Components of this Work's Objective
Chapter 3 - Experimental Analysis

Three sets of experimental work were performed in this study. The first set obtained the pressure drop across the airlift injectors. The second set determined the performance curve of the airlift pump system. The third set consisted of field work to study the operation of a raceway system by measuring the velocity profiles across the raceway. The first two sets were performed using a laboratory setup at the University of Guelph, while the last set consisted of field trials at an industrial aquaculture facility found in Ontario, Canada. The experimental work recorded from each set was used to validate the numerical solutions obtained from the simulations.

3.1 Laboratory Setup

The testing rig for the airlift pump consisted of various parts that allowed for the operation of the airlift pump under various conditions. An illustration of the setup is shown in Figure 3-1. The water from a reservoir tank was pumped into a secondary supply tank ① that was set at a certain height to maintain a water head, thus creating a specific submergence ratio for the airlift pump system.

The excess water pumped into the supply tank was diverted into the overflow section, which recirculated the water back into the water reservoir. The water from the supply tank filled the airlift pump ② along with the upriser tube ③ to simulate a constant submergence ratio.

The airlift pump was a dual injector type patented by Badr and Ahmed [2012]. A compressor provided the air supply, and the amount of air injected into the airlift was monitored through a flowmeter. The water was then pumped using the airlift pump through a clear acrylic upriser pipe with a $2.54 \times 10^{-2}$ m diameter and 1.6 m length. At the top of the pipe was a collection tank ④ that later dispelled the water into the measuring tank ⑤.
The measuring tank was marked in 1-liter increments to measure the collected volume of water. The water from the measuring tank was then allowed to return to the main water reservoir to complete the cycle for each experimental measurement case. A capacitance sensor 6 was attached to the upriser pipe and connected to a data acquisition system (DAQ) 7 and a computer interface 8 to measure the local void fraction.

A high-speed camera 9 was also used to record the flow structure which occurred inside the upriser pipe. The camera was capable of taking several hundred to thousands of frames per second, thus allowing the flow patterns and bubbles in the pipe to be tracked visually.
Figure 3-1 - Experimental Setup

<table>
<thead>
<tr>
<th>Number</th>
<th>Item</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Water supply tank</td>
</tr>
<tr>
<td>2</td>
<td>Airlift Pump -1.1 in</td>
</tr>
<tr>
<td>3</td>
<td>Riser Pipe</td>
</tr>
<tr>
<td>4</td>
<td>Collection Tank</td>
</tr>
<tr>
<td>5</td>
<td>Measuring Tank</td>
</tr>
<tr>
<td>6</td>
<td>DAQ</td>
</tr>
<tr>
<td>7</td>
<td>Computer</td>
</tr>
<tr>
<td>8</td>
<td>High-Speed Camera</td>
</tr>
</tbody>
</table>
3.2 Field Setup

An airlift pump system was installed at an aquaculture facility in McMillian Pitts, Ontario, Canada to study the effects of the pumping system on the operation of the raceway. The raceway system was built on a floating dock in a closed lake creating a recirculating system. A frame was built to fit the raceway and an airlift pump system was attached to it. This supplied the raceway with the needed operational flow rate. The airlift pump system was built out of PVC and consisted of four airlift pumps, each having a 1.016x10^{-1} m inner diameter. Figure 3-2 shows the airlift pump system build frame.

This system was then secured onto the raceway and the floating dock. The air supply was provided by an air blower that was present on site and was connected to each injector through the tubes present at the top. The airlift pumps lifted water from a certain depth in
the lake (1.78 m) to the surface, where it flowed through the raceway and exited through an opening at the back into the lake. Figure 3-3 shows the airlift pump setup as well as the raceway and the installation of the system.

Figure 3-3- Installation of the Airlift Pump System onto the Raceway

The purpose of the field work was to study the velocity profiles across and along the raceway, as well at various depths. The velocity profiles provided a baseline measurement which were then compared to the simulated results. These velocities were also important to the operation of the raceway, as a certain velocity was needed to aid in
fish growth. The measurement instrumentation used was an acoustic doppler velocimeter (ADV).

The ADV was attached to a flat plate to ensure that the orientation of the velocimeter probe was always correct and consistent. The flat plate was also attached to a roller that would go on a rail that was placed across the width of the raceway, as seen in Figure 3-4. Figure 3-5 provides a schematic of the locations where the velocity was measured. The grid provided suitable coverage to capture the effects of the airlift pump on the flow structure of the raceways. The readings were also taken at three different depths to further study the effects of the system on the flow at several levels.

Figure 3-4- Velocity Measurements Using ADV
Figure 3-5 - Experimental Velocity Measurement Schematic

- Measurements along the pump centerline
- Measurements in between the pumps
3.3 Instrumentation

Various measurement instrumentations were used to quantify the flow parameters being measured. This section will provide a small description of all the measuring instruments used in the experimental work.

3.3.1 Air Flow Meter

The OMEGA FMA-LP1600 gas mass flow meters were used to measure the air volumetric flow rate entering each injector of the airlift pump. These flow meters can be used to measure gas flows ranging from 0 to 500 SLM. The flow meters are also equipped with temperature and pressure sensors which allows for the measurement of these parameters as well. The flow meter was primarily used to measure the air volumetric flow going to each injector. The flow meter was also used to measure the pressure at the injector, as well as the air temperature flowing through the system. The flow meter was attached to an adjustable needle valve to control the amount of flow used. The pressure and temperature were used to calculate the mass flow for use in the CFD and validation.

3.3.2 Graduated Measurement Tank

A graduated tank was used to measure the amount of water pumped by the airlift pump in the laboratory setup. The measurement tank was connected by sloped tubes to the collection tank at the top of the upriser pump in the experimental setup. This ensured that all the water from the collection tank was transferred into the measurement tank. The measurement tank was built from acrylic and had a ball valve at the bottom of it. When measuring the water flow rate, the valve was closed to allow the water to fill up, while a timer was used to find the amount of time required to fill up the tank. The tank was marked at 1L intervals.

3.3.3 Capacitance Sensor

A capacitance sensor is a device that measures the local void fraction using the dielectric properties of air and water. The capacitance sensor consists of two or more copper plates
that are connected across from each other, effectively creating a capacitor. This sensor is placed around a pipe where the fluid inside the pipe acts as the dielectric material between the capacitor. Since air and water have different dielectric values, the voltage stored across the sensor will change, effectively changing the capacitance.

The capacitance value obtained from each phase was collected and used to calibrate the capacitance sensor. A correlation between capacitance and void fraction was established. When the airlift was operational and two-phase flow occurred in the upriser pipe, the sensor was able to detect the amount of each phase flow based on the capacitance reached across the capacitor and convert it to a void fraction percentage. The capacitance sensor was ideal as it was nonintrusive and therefore did not disturb the flow inside the pipe.

3.3.4 Acoustic Doppler Velocimeter (ADV)

An ADV is a device that measures the instantaneous velocity of fluids by measuring the Doppler shift experienced by ultrasound radiation. A FlowTracker Handheld ADV was used with a 2D measurement configuration to measure the normal and tangent velocity components of the flow. The device consisted of a single acoustic transmitter and two or more receivers depending on the measurement configuration. The transmitter generated a short pulse of sound at a known frequency, which was reflected in all directions by the particulate matter in the water. The receivers were then able to pick up the reflected noise [136]. The velocimeter then measured the change in frequency at each receiver and used the Doppler shift principle to find the equivalent velocity.
Calibration and Uncertainty

In the laboratory, the test setup used two identical mass flow meters. These meters had a ±0.8% reading uncertainty as well as a ±0.2% full scale uncertainty. Pressure measurements were also obtained from the mass flow meter with a reading error of ±68.9 Pa which equated to ±8% of the value. The volumetric water flow was recorded using a measurement tank, which was subdivided into one-liter increments, as well as a stopwatch with one millisecond intervals. Since these measurement instruments depend highly on human interpretation, the uncertainty associated with the water flow rate was calculated to be ±5%.

The capacitance sensor was calibrated by draining the water from the system, and recording the capacitance reading of the air phase once the water had dried in the pipe. Similarly, when the system was filled with water, the capacitance value was recorded for the water phase. The difference in capacitance was typically in the order of 10 pico-farads. A linear correlation between void fraction and capacitance was established to be used when the system was operational. The capacitance sensor was observed to have a repeatability uncertainty of ±2.5%
The ADV was calibrated by placing the probe in stagnant water and ensuring that the output velocity was zero. The ADV reading uncertainty is typically ±0.01 m/s, which, based on the magnitude of the velocities experienced, is nearly 15%, and it is expected to an increase in more turbulent flow regions. The ADV also has a ±1% uncertainty in the speed of sound signal, which results in an additional ±2% uncertainty in the velocity measurement. The velocimeter in the field was subjected to both turbulent and highly recirculating flows at the points closest to the airlift pump system, and so the uncertainty range was expected to be higher.

Table 3-1- Summary of Instrumentation Uncertainties

<table>
<thead>
<tr>
<th>Instrument</th>
<th>Unit</th>
<th>Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow Meter</td>
<td>$\frac{L}{\text{min}}$</td>
<td>±1.2%</td>
</tr>
<tr>
<td></td>
<td>Pa</td>
<td>±8%</td>
</tr>
<tr>
<td>Capacitance Sensor</td>
<td>pF</td>
<td>±2.5%</td>
</tr>
<tr>
<td>ADV</td>
<td>$\frac{m}{s}$</td>
<td>±17%</td>
</tr>
<tr>
<td>Graduated Measuring Tank &amp; Stop Watch</td>
<td>$\frac{L}{s}$</td>
<td>±5%</td>
</tr>
</tbody>
</table>
Chapter 4 - Numerical Analysis

Computational fluid dynamics (CFD) is a branch of fluid mechanics that utilizes numerical simulations to study fluid dynamics of internal or external flows. The use of CFD has been on the rise as computational models have become more thorough. Coupled with the recent advances in modern CPU processing, this has allowed for evermore complex and elaborate simulations to be performed. A commercial simulation package (FLUENT 19.0) developed by ANSYS INC. was used to perform the numerical simulations for all the modelling present in this work.

The use of CFD can now be found in nearly all engineering applications. Using it without understanding the mathematical underpinnings of the computational models at work will result in disastrous and nonsensical solutions. It is also important to note that many simulations also need additional numerical modelling to better represent the physical flow field. Turbulence models are examples of the additional numerical work required to represent turbulent flows. The simulation of multiple phases also requires special numerical formulation to account for interactions between the phases. This chapter will cover the mathematical background of the computational models used in this work.

4.1 Airlift Injector in Single Phase Conditions

The cornerstones of all numerical modelling in CFD are the Navier-Stokes equations, as they describe the continuity, momentum and energy movement in any flow domain. These equations, along with Euler’s approach to fluid flow, provide a framework to solve fluid dynamic problems using the control volume method [137]. In all flow simulations, a geometry is divided into smaller areas, where each area represents a control volume. At each control volume, the applicable parts of the Navier-Stokes equations are solved in order to obtain an approximate solution for the flow field. The time dependent continuity equation is presented as:
The continuity equation ensures that the amount of mass entering a control volume is equal to the amount leaving the volume plus any additional mass that may have originated from within the control volume. This equation satisfies the mass conservation law and is applicable for incompressible and compressible flows. The time dependent momentum conservation equation is presented as:

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

(2)

The momentum conservation equation ensures that Newton's second law is preserved, as the total force being applied on a control volume is equal to its mass times the acceleration of the volume. There are two main forces that act on a control volume: body forces and surface forces. The effect of gravity as well as any external forces applied to the control volume are the main body forces observed. Surface forces originate from the pressure distribution acting on a surface and the stresses that occur due to friction between the fluid surface and other bodies [138]. Shear and normal stresses are the main surface forces observed and are described in Equation (5). The energy equation is presented as:

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\mathbf{v} (\rho E + p)) = -\nabla \cdot \left( \sum_j h_j J_j \right) + S_h
\]  

(5)

The energy equation ensures that the first law of thermodynamics is preserved. It ensures that any heat flux that is added to the control volume is equal to the rate of work done on the element due to body and surface forces. Depending on the flow problem assumptions,
the energy equation may not be needed to provide a solution to the flow domain, such as in the case of adiabatic flows.

Navier-Stokes equations are very useful, as they describe the physical phenomenon of fluid dynamics, but unfortunately due to their high complexity they cannot be solved analytically except for very idealized cases. Thus, CFD merely provides an approximate solution to these equations. However, in well-defined cases with appropriately chosen parameters, these approximations are usually accurate enough to be used in most design applications.

4.2 Airlift Pump Modelling in Two-Phase Flow Conditions

Multi-phase flow is very different than single phase flow; multiple cases could exist at the same flow rates within a multi-phase system where each case could be modeled differently. Regardless of the flow experienced, the governing equations of conservation of mass, momentum and energy are still applicable. The formulation of these equations changes based on the nature of the two-phase flow experienced in the problem. The homogenous, separated flow, and drift flux are all analytical models used to describe specific flow conditions. In an airlift pump, the flow has been observed to undergo several flow patterns which affect the performance of the system; modelling each phase separately is therefore more appropriate. The general separated flow model is described to provide some familiarity with analytical two-phase flow modelling. Assuming a two-phase flow system ($q$ and $p$), two conservation of mass equations would be formulated as follows:

$$\frac{\partial}{\partial t} \left[ \rho_q (1 - \alpha) \right] + \nabla \cdot \left[ \rho_q (1 - \alpha) \mathbf{v}_q \right] = \dot{m}_{qp} + S_q$$ (6)

$$\frac{\partial}{\partial t} \left[ \rho_p \alpha \right] + \nabla \cdot \left[ \rho_p \alpha \mathbf{v}_p \right] = -\dot{m}_{qp} + S_p$$ (7)

Conservation of mass considers any mass transfer occurring between the two-phase flows, as well as any external sources that might generate any mass. These source terms,
however, are almost invariably equal to zero [48]. The conservation of momentum also has two equations representing each phase separately. They are formulated as such:

\[
\rho_q \left( \frac{\partial \vec{v}_q}{\partial t} + \vec{v}_q \cdot \nabla \vec{v}_q \right) = \vec{b}_q + \vec{f}_q - \nabla p \tag{8}
\]

\[
\rho_p \left( \frac{\partial \vec{v}_p}{\partial t} + \vec{v}_p \cdot \nabla \vec{v}_p \right) = \vec{b}_p + \vec{f}_p - \nabla p \tag{9}
\]

The momentum equations include the body forces applied per unit volume as well as any other forces applied on the control volume. These forces are dependent on the area of interest that the model is attempting to capture, and so they can be as simple or as complex as need be. The energy equation for two-phase flow using the separated model is much more complex and is very dependent on the flow system. The most general form of the energy equation for each phase is presented as:

\[
\frac{\partial}{\partial t} \left( \alpha_q \rho_q h_q \right) + \nabla \cdot \left( \alpha_q \rho_q \vec{v}_q h_q \right) = \alpha_q \frac{dp_q}{dt} + \vec{i}_q + S_q + \sum_{p=1}^{n} \left( \dot{m}_{pq} h_{pq} - \dot{m}_{qp} h_{qp} \right) \tag{10}
\]

\[
\frac{\partial}{\partial t} \left( \alpha_p \rho_p h_p \right) + \nabla \cdot \left( \alpha_p \rho_p \vec{v}_p h_p \right) = \alpha_p \frac{dp_p}{dt} + \vec{i}_p + S_p + \sum_{p=1}^{n} \left( \dot{m}_{qp} h_{qp} - \dot{m}_{pq} h_{pq} \right) \tag{11}
\]

The energy term could include the internal energy of each phase, the heat transfer, as well as the work done on the phases due to pressure, body forces, shear or any other applied work. This equation, depending on the analysis type, could be simplified or made more complex. The governing equations are hardly ever used in their general form; simplified models are typically created to suit an individual case, such as inertial
dominated flows, phase changes, heat and mass transfer in boiling and cooling, and many others.

In two-phase flow numerical simulations, the introduction of a new phase into the flow domain also adds further challenges to numerically modelling the flow. These challenges are mainly present due to the difficulty in modelling the pressure domain, and the flow structure of the interacting phases on a computational grid. Similarly, multiple CFD models have been developed to represent multi-phase flow, each with its own level of accuracy, solution variables, and suitable multiphase flow application. The models either use an Euler-Lagrange or an Euler-Euler approach to create a specific reference frame and coordinate system to setup the solution’s formulation. The Euler-Lagrange approach assumes that the fluid phase is a continuum by solving the time averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of bubbles and particles in the flow domain [139]. This approach is suitable for very bubbly flow, dispersed flow such as spray flows, and any flows in which the volume fraction of the second phase could be assumed to be negligible. In an airlift pump system, which has a flow in the upriser pipe that is affected by the volume fraction and the changing flow patterns, this approach is not suitable and was not used. The Euler-Euler approach treats each phase mathematically as an interpenetrating continuum [139]. This approach adds to the assumption that no two-phases can occupy the same volume. The phasic void fraction is introduced as variable in the solution process. This results in the creation of additional conservation equations for each phase present in the flow domain. This makes models that have been developed using a Euler-Euler approach suitable to be used for simulating the operation of airlift pumps.

Three major Euler-Euler approach models are used in multi-phase simulations: volume of fluid (VOF), mixture, and Eulerian models. Each of these models is suited for a specific application. The approach to the solution also varies from one model to the other. Volume of fluid and mixture models are part of the one equation family, in which one equation is formulated and solved for the conservation of momentum and energy. The volume of fluid
model differs from the mixture model regarding the conservation of mass formulation, in that the VOF model creates a mass equation for each phase present in the system. The Eulerian model on the other hand is much more powerful, as it formulates all parts of the governing equations for each phase in the system.

The VOF model with its strong interface tracking ability is used for two-phase flow cases in which the motion of large bubbles, separated flows or free surface flows are of interest. The mixture model formulates the governing equations in terms of the relative velocity between the two phases, which makes it a desirable model for dispersed and homogeneous flows. The Eulerian model is capable of modelling most flow conditions but suffers from slow computational time due to the high intensity of solving the complete governing equations. The mixture and the Eulerian models both suffer from a lack in accuracy when attempting to follow the surface interfaces between the fluids, and so cannot produce accurate representations of the void fraction in the flow field. The Eulerian model can be coupled with the VOF to allow for volume fraction tracking while solving the complete governing equations. This method adds further to the computational load of the Eulerian model but is capable of producing a more detailed solution of the flow field.

Chapter 2 has shown that the two models mostly used in similar work are the Eulerian and the VOF models. The purpose of the current work is to model the performance curve of the dual injector airlift pump system, while studying the flow patterns occurring in the upriser pipe, and performing a free surface simulation of an aquaculture raceway. The most appropriate model for this work is therefore the VOF model. This model permits a study of the phase interactions and flow patterns in the upriser pipe, while also following the water level and the waves generated in the raceway. The operational cases for both simulations are well suited for the VOF model. All the information required to complete this study, such as the velocity field in the raceway and the performance of the airlift, can be obtained using the VOF without the need for a more computationally intense model, such as the Eulerian-VOF.
<table>
<thead>
<tr>
<th>Model Type</th>
<th># of Continuity Equations</th>
<th># of Momentum Equations</th>
<th># of Energy Equations</th>
<th>Applicable Flow Regime</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume of Fluid</td>
<td>n fluids</td>
<td>1</td>
<td>1</td>
<td>Slug, Stratified, Separated, Surface flow</td>
</tr>
<tr>
<td>Mixture Model</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>Bubbly, Droplet, Slurry</td>
</tr>
<tr>
<td>Eulerian</td>
<td>n fluids</td>
<td>n fluids</td>
<td>n fluids</td>
<td>Bubbly, Droplet, Separated, Slurry</td>
</tr>
</tbody>
</table>
4.2.1 Volume of Fluid Model (VOF)

The VOF model is a surface tracking technique developed with an Euler-Euler approach. The main advantage of this model is its ability to track and present the interface between the distinct phases in the flow domain. With a VOF model, a continuity equation is used for each phase in the domain, while a single set of momentum equations is shared by the fluids. The void fraction is tracked throughout the domain [139]. In each control volume the total void fraction of all the phases must be equal to one. All the variables calculated in the flow domain are shared by the phases. These phases represent volume averaged values, as long as the void fraction in each control volume is known [140]. In any control volume, if two-phases are present (q and p) the following three conditions are possible:

\[ \alpha_q = 0 \quad \text{Cell is empty of the q}^{th} \text{ fluid, full of p}^{th} \text{ fluid} \]  
\[ \alpha_q = 1 \quad \text{Cell is full of the q}^{th} \text{ fluid, empty of p}^{th} \text{ fluid} \]  
\[ 0 < \alpha_q < 1 \quad \text{Cell contains an interface between p}^{th} \text{ and q}^{th} \text{ fluid} \]

The tracking of the interface occurs when solving the continuity equation for one of the phases, and so the continuity equation for the tracked phase (q) is presented as:

\[
\frac{1}{\rho_q} \left[ \frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) \right] = \sum_{p=1}^{n} \left( \dot{m}_{pq} - \dot{m}_{qp} \right) S_{\alpha_q} 
\]

Equation 15 is also called the volume fraction equation. The non-tracked phase (p) continuity equation is presented as:

\[
\frac{\partial}{\partial t} (\alpha_p \rho_p) + \nabla \cdot (\alpha_p \rho_p \vec{v}_p) = \sum_{p=1}^{n} (\dot{m}_{qp} - \dot{m}_{pq}) S_{\alpha_p} 
\]
The interface between the two fluids is not defined by uniform or easily predictable shapes. This may be seen in the operation of airlift pumps, in which the flow undergoes several flow pattern changes. This makes it difficult to model the interface of the phases at partially covered control volumes. An interpolation method is needed near the interface areas. At this point the convection and diffusion fluxes at the control volume faces need to be computed and balanced with the source term within the control volume itself [140]. The compressive scheme was chosen to perform the void fraction interpolation. The compressive scheme developed by Ubbink [141] is a second order reconstruction scheme based on a slope limiter. The slope limiter is a numerical scheme used to limit the solution gradient, discontinuities or sharp changes in the flow domain that occur in high order spatial discretization schemes. The compressive scheme is presented as:

$$\phi_f = \phi_d + \beta \nabla \phi_d$$  
(17)

where the value of the slope limiter is directly related to the discretization scheme used. These values are summarized as:

$$\beta = 0$$  
First order upwind scheme  
(18)

$$\beta = 1$$  
Second order schemes  
(19)

$$\beta = 2$$  
Compressive scheme  
(20)

$$0 < \beta < 1 \text{ and } 1 < \beta < 2$$  
Between 0 and 1 means blending of the first order and the second order. Between 1 and 2 means blending of the second order and the compressive scheme.  
(21)

This scheme allows for a good multi-phase flow structure prediction and is able to accurately model the interaction between the phases.

As mentioned before, only a single momentum equation is solved throughout the domain. The velocity field generated by solving this equation is shared between the phases and
is dependent on the volume fraction and the shared density and viscosity. The momentum equation along with the shared density formulation is presented as:

\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot [\mu (\nabla \vec{v} + \nabla \vec{v}^T)] + \rho \vec{g} + \vec{F} \tag{22}
\]

\[
\rho = \alpha_q \rho_q + (1 - \alpha_q) \rho_p \tag{23}
\]

The energy equation is also shared among the phases, and so the energy and temperature values are presented as mass averaged variables. The energy equation in the VOF model is presented as:

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot (k_{eff} \nabla T) + S_h \tag{24}
\]

\[
E = \frac{\sum_{q=1}^{n} \alpha_q \rho_q E_q}{\sum_{q=1}^{n} \alpha_q \rho_q} \tag{25}
\]

In the current work, the energy equation was not used in the solution method as the flow was assumed to be adiabatic and therefore no heat or mass transfer is present. These equations are the building blocks of all multi-phase numerical simulations utilizing the VOF model. In many engineering problems, these equations might not be sufficient to accurately model complex flows, and so additional modelling is required to account for the flow conditions.

### 4.3 Turbulence Parameters

Turbulent flows are heavily encountered in almost all engineering applications, yet modelling these flows is very complex. All numerical modelling of turbulent flow approximates the various turbulent phenomenon occurring, while focusing on one parameter that is of interest to the application. The flow in the dual airlift injector is considered turbulent due to the highly complex geometry that the flow must undergo. This generates vortices and eddies that further disturb the flow even at low speeds. Two-phase flow in an airlift pump system further complicates the flow structures; it is therefore
considered turbulent even at lower velocity cases. This turbulence needs to be accounted for to better study the performance and flow structure of the flow inside the airlift pump.

Numerous models have been developed to describe turbulent flows numerically. The selection of an appropriate turbulence model depends on the expected physics of the flow, the established practices, the level of accuracy required, and the available computational resources. The works of Tabib *et al.* [95] and Ekambara and Dhotre [96] found that Reynolds averaged Navier-Stokes turbulence models, such as K-ε and K-ω, provided adequate predictions while being computationally economical. It was also determined that the large eddy simulation model provided more accurate results, but the additional computational time required to complete the simulation could not justify any gains in accuracy. This makes the K-ε and K-ω turbulence models the logical choice for the purpose of this work.

4.3.1 Reynolds Averaged Navier-Stokes Equations

Reynolds averaged Navier-Stokes equations (RANS) are the time averaged equations of fluid flow equations. These equations are derived using the Reynolds decomposition theory. In this theory, an instantaneous quantity is decomposed into its time averaged and fluctuating quantities [142]. All scalar quantities such as pressure and velocity would be presented as:

\[
\phi = \bar{\phi} + \phi'
\]  

(26)

Substituting this form into the Navier-Stokes equations would result in the RANS equations:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho}{\partial x_i} (\rho u_i) = 0 \tag{27}
\]

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j)
= - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right]
+ \frac{\partial}{\partial x_j} \left( -\rho \ddot{u}_i \ddot{u}_j \right) \tag{28}
\]

These equations have the same form as the instantaneous Navier-Stokes equation, but the variables are now presented as ensemble or time averaged values. The RANS momentum equation (28) has an additional term that represents the effects of turbulence, which are the Reynolds stresses. This additional Reynolds stress term must be solved in order to complete the calculation of the RANS equations. This creates a closure problem when completing the RANS equations; therefore, a system of equations is needed to complete the calculation process and close the turbulence problem.

1) K-\(\epsilon\) Turbulence Model

The K-\(\epsilon\) turbulence model is a two-equation model that allows for the determination of a turbulent length and time scales to be solved. This model was developed by Launder and Spalding [143], and has become the most popular turbulence model used in numerical simulations due to its robustness, reasonable accuracy, economic computational intensity and its ability to be applied to a wide range of turbulent flows [140]. The model solves the transport equation of the turbulence kinetic energy and its dissipation rate [143]. This model assumes that the flow is fully turbulent and so it is best used in such cases. The turbulence kinetic energy (k) and rate of dissipation (\(\epsilon\)) are obtained from the following transport equations [140]:

64
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) \\
= \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M \\
+ S_k
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) \\
= \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{1\varepsilon} G_b) \\
- C_{2\varepsilon} \rho \frac{\varepsilon}{k} + S_\varepsilon
\]

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]

\[
C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3
\]

The values for the constants are based on experiments of turbulent flows including frequently encountered shear flows at the boundaries, as well as for decaying isotropic grid turbulence [140].

2) K-\( \omega \) Turbulence Model

The K-\( \omega \) turbulence model is also a two-equation model, which analyzes turbulent flow using the turbulence kinetic energy and the specific dissipation rate. This model, which was developed by Wilcox [144], accounts for modifications for low-Reynolds number effects, compressibility and shear flow spreading. The model has been modified over the years to improve the accuracy of predicting free shear flows, while reducing its sensitivity to the effects of the turbulent kinetic energy and dissipation values outside of the shear layer [140]. The turbulence kinetic energy \( (k) \) and specific dissipation rate \( (\omega) \) are obtained from the following transport equations [140]:

65
\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k + S_k \quad (33)
\]

\[
\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + S_\omega \quad (34)
\]

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad (35)
\]

\[
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \quad (36)
\]

\[
\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (37)
\]

In the case of a low Reynolds number turbulent flow, a corrective term is added to the turbulent viscosity, which is given by:

\[
\alpha^* = \alpha_0^* \left(\frac{\alpha_0^* + Re_t / R_k}{1 + Re_t / R_k}\right) \quad (38)
\]

\[
Re_t = \frac{\rho k}{\mu \omega} \quad (39)
\]

\[
\alpha_0^* = \frac{\beta_i}{3} \quad (40)
\]

\[
R_k = 6, \beta_i = 0.072 \quad (41)
\]
4.4 Geometry and Meshing

Geometry modelling and meshing are the initial steps of any numerical simulation. A graphical model needs to be created that best represents the physical flow domain, while not being too complicated to create solution issues.

4.4.1 Geometry and Boundary Conditions

In total, there were three models used for this work: the dual airlift injector, the airlift pump system, and the aquaculture raceway models. Each of these models was used to study certain parameters and flow structures, and they were later combined to provide a complete picture of the problem and the predicted solution. All the work completed utilized 3D models for the simulation. Numerically, the Navier-Stokes equations are boundary value problems, and so, boundary conditions must be defined to mathematically describe the operation of the system. In CFD, boundary conditions indicate which areas of the model are inlets, outlets or the numerous other boundary conditions available. In this work the colors used to identify the boundary conditions are as follows:

- Green → Inlet
- Red → Outlet
- Yellow → Symmetry
- Grey → Solid Wall

4.4.1.1 Dual Injection Airlift

The dual injection design of the airlift was developed by Ahmed and Badr [67] as a way to improve the efficiency of a traditional air injector in an airlift pump. The design involved two injection strategies, an axial and a radial as can be seen in Figure 4-1. The axial injector pushes the air through an inner annulus to redirect its movement into the axial direction. The radial injector consists of a perforated tube that provides air in the radial direction.
In order to use the model for a numerical simulation, the inner parts of the injectors where the flow occurs are modelled. Figure 4-2 shows the 3D model developed for the case with its dimensions.
It can be seen that the injector consists of three distinct sections: the axial and the radial injectors, and the inner pipe that would be connected to the upriser and suction pipes. The flow in the injector was assumed to be radially symmetrical along the core of the inner pipe. Only a quarter of the airlift model was therefore needed to conduct the simulation. This simplified the simulation work and reduced its computational intensity. The boundary conditions for this model are presented in Figure 4-3.
At the bottom of the injector an inlet boundary condition is also present but is blocked by the view in Figure 4-3. These boundary conditions accurately represent the physical attributes of the dual injector. The specific values of the boundary conditions selected are discussed in Chapter 5.

### 4.4.1.2 Dual Injection Airlift Pump System

This system, which was comprised of a dual injector with an upriser tube, completed the airlift pump system setup. The system was built to replicate the experimental setup found in Chapter 3. Figure 4-4 shows the setup of the airlift pump system with its dimensions.
With the scale of the new model, the system now consisted of only two parts: the dual injection airlift pump and the upriser tube. The symmetrical flow assumption was also applied in this simulation and a quarter of the domain was needed. The boundary conditions of this setup are presented in Figure 4-5.
Similarly, an inlet boundary was present at the bottom of the domain but is blocked by the view in Figure 4-5. These boundary conditions accurately represent the physical attributes of the experimental setup found in Chapter 3. The specific values of the boundary conditions selected are discussed in Chapter 5.
4.4.1.3 Aquaculture Raceway

The raceway was modelled based on an airlift pump system found in at McMillian Pits facility in Ontario, Canada. Figure 4-6 shows the setup of the raceway along with the airlift pump system.

Figure 4-6- Aquaculture Raceway Model

Four airlift pumps were used in this setup consisted each had an inner diameter of 1.016x10^{-1} m. To simplify the simulation geometry and reduce the numerical intensity of the model, only one section of the raceway was used for the simulation, while the inlet geometry was changed into a square based on the hydraulic diameter of the airlift pump system. Since the simulation was primarily focused on the interaction of the air-water mixture exiting the airlift pump into the raceway, the CFD model had to include certain zones. The airlift pump exit was located above the water level in the raceway. This
required the addition of an air zone that represented the atmosphere above the raceway. The resulting geometry was a 3D rectangular model that was divided into various regions to facilitate better meshing of important regions. Figure 4-7 shows the geometry and dimensions of the raceway model that was used for the simulation.

Now that the model was more defined, it was important to identify the air and water bodies present in the domain. The water domain was selected based on the dimension of the raceway, as that is where the water was present. The air domain was defined as any area above the raceway.

Figure 4-8 indicates the boundary conditions of the raceway model. As previously discussed, the model had to include a water zone which represented the raceway, along with an outflow boundary to allow the water to exit. The air zone also had to contain an outflow boundary to allow for the added air to escape. The inlet was located in the air
zone and channeled an air-water mixture into the water zone. Since this was only a slice of the raceway, two symmetry boundaries were applied to both sides of the model.

Figure 4-8- Boundary Conditions for the Raceway Domain

While this is a simplified representation of the physical raceway, the model geometry and boundary conditions applied represented a very good approximation of the physical phenomenon present. The simulation setup was sufficient to study the parameters of interest for this work.

4.4.2 Meshing

Meshing is the process of creating the finite control volumes in the flow domain. The meshing process has an effect on the stability and accuracy of any fluid simulation and
thus requires a great deal of attention and care. The meshing process utilized the built-in meshing tool in the ANSYS Multi-Physics package.

4.4.2.1 Dual Injector and Airlift Pump System

The meshing process for the dual injector and the airlift pump system followed the same guidelines due to the resemblance in the geometry. The dual injector was meshed using a structured non-conformal mesh due to the complex geometries presented. The geometry was divided along every major change of geometry in the model. These cuts allowed for better control when meshing, while also made it possible to achieve the appropriate structure of the mesh.
It is known that turbulent flow will be experienced in the injector and the pump system; consequently, some areas were identified for their high impact on accuracy of the simulation and its solution. The flow at the boundaries along the walls and through the axial and radial injectors are of high importance, and special meshing requirements are needed at such locations. In order to sufficiently capture the flow through these areas, a mesh resolution between 10 and 15 nodes at the wall boundary layers is desirable[140]. Such requirements could be evaluated by analyzing the performance of the wall functions.
by using the $y^+$ parameter. This parameter is a non-dimensional number that is numerically defined as:

$$y^+ \equiv \frac{u_* y}{v} \quad (42)$$

$$u_* \equiv \sqrt{\frac{\tau_w}{\rho}} \quad (43)$$

$$\tau_w = \mu \left( \frac{\partial u}{\partial y} \right)_{y=0} \quad (44)$$

The parameter helps to indicate the thickness of the first layer based on the surface velocity, the kinematic viscosity and the length of the effected domain. Based on the turbulence model chosen and the geometry of injector, a $y^+$ value of 15 was deemed sufficient and the thickness of the first layer was determined. The rest of the model was optimized by using a bias. This increases the quantity of the elements at the areas of interest, while using a more conservative number of elements in other areas. Figure 4-10 illustrates the meshing of the pump as well as some locations on the upriser pipe.
Figure 4-10- Dual Injector Airlift Pump System Meshing
4.4.2.2 Aquaculture Raceway

The meshing for the raceway was simplified due to the simplified geometry of the case. The geometry was also divided to allow for finer meshing in the areas of interest. A conformal and structured quadrilateral mesh was applied to the model. Special care was applied to areas where air-water interactions were experienced, such as the water inlet, the air-water interface area (i.e. the area next to the inlet), and the water-air surface (i.e. near the raceway outlet).

![Raceway Domain Mesh: Side View (Top), Front View (Bottom)](image)

Figure 4-11- Raceway Domain Mesh: Side View (Top), Front View (Bottom)
Figure 4-11 presents the meshed domain of the raceway. It can be seen how the meshing technique was applied to each zone in the domain. The areas with the highest mesh density can also be seen.

4.4.2.3 Meshing Summary

The following table (4-2) presents the number of elements and nodes used for each model in the simulation. Along with the numbers, some quality parameters are also presented.

<table>
<thead>
<tr>
<th>Model</th>
<th># of Elements</th>
<th># of Nodes</th>
<th>Maximum Skewness</th>
<th>Maximum Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dual Injector</td>
<td>236,862</td>
<td>274,529</td>
<td>80% of elements with 0 skewness</td>
<td>8</td>
</tr>
<tr>
<td>Dual Injection Airlift Pump</td>
<td>292,225</td>
<td>341,243</td>
<td>88% of elements with 0 skewness</td>
<td>12</td>
</tr>
<tr>
<td>Raceway</td>
<td>322,512</td>
<td>342,940</td>
<td>0</td>
<td>6</td>
</tr>
</tbody>
</table>

4.5 Discretization and Solution Method

Discretization refers to the numerical solution methods used to calculate the solution from the domain. A pressure-based solver was chosen as the algorithm to solve the domains presented where the scheme employed a projection method in which the continuity and velocity fields in the domain were achieved through solving a pressure equation [145]. The solution process of these schemes required an iterative approach, since all the governing equations were coupled to one another. The SIMPLE scheme which was developed by Spalding and Patankar [146] was chosen from the various pressure-based schemes available. This method was chosen as it is very suitable for both single and multi-phase simulation work. The scheme was applied by constantly calculating the pressure and velocity fields while checking the validity of the continuity equation. Each time a cycle occurred, a correction factor for the pressure and velocity fields was applied.
Once the correction between each cycle became small enough, it indicated that the solution was reaching convergence and thus ended the calculation cycle. In a steady state simulation, the results obtained after convergence would be the finalized solutions. While in a transient simulation, this would indicate the jump to the next time step in the process. The convergence criteria for the continuity, velocity components and turbulence component were set at $1 \times 10^{-5}$ for all the single-phase flow simulations. This selection ensured that the residuals would be minimized to ensure true convergence of the solution. The convergence criteria for the two-phase flow work was set at $1 \times 10^{-3}$. This is due to the addition of an extra residual term that accounts for the void fraction of the phases, thus making more difficult to achieve much smaller residual criteria. The complexity of the flow that occurs in the two-phase flow simulations also makes difficult for the solution to converge on any tighter parameters than the ones chosen. Overall, the convergence criteria chosen for the simulation work fits the scope and aims of the numerical work.
Figure 4-12- SIMPLE Discretization Algorithm
Each of the variables solved use separate differencing schemes. Differencing schemes refer to the numerical method that is utilized to find the values at each node and element in the domain. Lower order differencing schemes are less accurate but are better for convergence, while the higher orders provide more accurate results but might lead to convergence issues. When running a transient simulation, a temporal differencing scheme is also required for each time iteration. A summary of the differencing schemes is presented in Table 4-3.

Table 4-3- Differencing Schemes Used

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Differencing Scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>PRESTO!</td>
</tr>
<tr>
<td>Momentum</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Volume Fraction</td>
<td>Compressive</td>
</tr>
<tr>
<td>Turbulence</td>
<td>First Order Upwind</td>
</tr>
<tr>
<td>Transient</td>
<td>First Order Implicit</td>
</tr>
</tbody>
</table>
4.6 Summary of Assumptions

While the three components of this work come together to provide a complete picture of the ability of an airlift pump to operate an aquaculture raceway, each component studied had different assumptions to focus on predicting the parameters of interest. This section will provide a summary of the assumptions used for each simulation.

1) Dual Injector System

- Single Phase
- Steady state
- Incompressible flow as Mach number $<< 0.03$
- Adiabatic flow: No heat or mass transfer
- Turbulent flow: due to flow in complex geometry
- Flow is symmetric along the core of the airlift
- No added sources

2) Dual Injector Airlift Pump System

- Two-phase flow: Volume of Fluid Model
- Adiabatic flow: No heat transfer
- No mass transfer
- Turbulent flow
- Flow is symmetric along the core of the airlift
- No added sources

3) Aquaculture Raceway

- Two-phase flow: Volume of Fluid Model
- Adiabatic flow: No heat transfer
- No mass transfer
- Inlet shape has no effect on the flow
- Only one phase can exist in the same volume
- Flow is symmetric on both sides of the domain
- No added sources
Chapter 5 - Results and Discussion

This chapter will present and discuss the data acquired numerically and experimentally for each component of this study. Understanding how each part of the system operates gives a better understanding on how the system works together as a whole. Each component was analyzed and validated with its experimental counterparts. The first part of the system involved studying the hydrodynamics of the dual injection airlift, followed by a two-phase simulation of the airlift pump system, in which the performance of the pump was studied. Finally, the aquaculture raceway was studied based on the performance provided by the simulation of the airlift pump system.

5.1 Mesh Independence Study

A mesh analysis study was performed to ensure that the quality of the mesh used in the simulation was adequate. This was done by solving the problem employing multiple element numbers that were used to mesh the flow field. If the solution values were dependent on the number of elements used, this indicated that the mesh quality was not yet sufficient. Once a uniform solution was reached across multiple sets of element numbers, the solution became independent of the mesh quality, and the simulation process could proceed. For each model used, a separate mesh independence study was conducted. Only the solutions involving the relevant parameters were compared in each mesh case.

5.1.1 Dual Injection Airlift

Four different mesh cases were tested to study the effect of the number of elements used on the pressure and the velocity profiles. The number of elements used were as follows: 81,549; 149,660; 205,908; and 310,238. These four cases provided a solid understanding of how the element count affected the solution. The boundary conditions for this study were as follows:
Table 5-1- Summary of Boundary Conditions – Airlift Injector Mesh Independence Study

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Units</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Inlet &amp; Radial Inlet</td>
<td>Air Mass Flow Inlet</td>
<td>$\frac{kg}{h}$</td>
<td>94.75</td>
</tr>
<tr>
<td>Suction Inlet</td>
<td>Pressure Inlet</td>
<td>Pa</td>
<td>0</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Outlet</td>
<td>Pa</td>
<td>0</td>
</tr>
</tbody>
</table>

The values of the boundary conditions were arbitrarily chosen but were within the operational range of the airlift injector. A steady state simulation was conducted to obtain the solution of the flow domain. The area of most interest was the symmetry boundary that contained the air inlet, as it provided a full interaction between the inlet piece and the two injector types, as well as their effect on the main flow section. Figure 5-1 shows a side by side comparison of the contour plots of static pressure (a) and velocity (b) at the symmetry boundary for all four cases.
<table>
<thead>
<tr>
<th># of Elements</th>
<th>81,549</th>
<th>149,660</th>
<th>205,908</th>
<th>310,238</th>
</tr>
</thead>
</table>

(a)
Figure 5-1 - Contour Plots of (a) Static Pressure and (b) Velocity

Looking at the pressure contour plots from Figure 5-1(a) for all cases, it seems clear that the quality of the solution was initially affected by the number of elements but only marginal gains were realized thereafter. The data shows that if the number of elements is around 80,000 or less, the solution quality was not acceptable and mesh independence was not reached. Looking at the three cases with element counts >149,660, the solution remained constant with some smoothing at the contour edges. This smoothness was directly related to the increase in mesh density in the domain; thus, the solution values at each element were more distinct as were visualized in the contour plot. The velocity contour plots from Figure 5-1(b) show that the solution of the velocity field was less sensitive to the mesh density in the domain. The velocity magnitude and general distribution was essentially unchanged. These results suggested that an element number higher than 150,000 elements is required to achieve results that are mesh independent.
As discussed in Chapter 4, the final element number used for the numerical simulation was 274,529 elements. While this is higher than what is shown to be the minimum number of elements, this mesh also suited the turbulence requirements to capture the wall boundaries.

5.1.2 Airlift Pump System

The studies done on the dual injector showed the minimum number of elements required to achieve an independent mesh solution. However, with the addition of a 1.45-meter-long upriser pipe and the simulation of two-phases instead of one, a new mesh analysis was required. The domain was meshed using various number of elements: 114,588; 159,552; 215,030; 260,781; 287,237; and 349,316. The parameter of importance in these simulations was the airlift performance curve, which is created by obtaining the water flow rate exiting from the pump system compared to the amount of air injected. A steady state simulation was performed in each case to obtain the performance curve. While two-phase flow is a transient phenomenon, the steady state solution does not represent the physical system; it can, however, be used for the purpose of studying the mesh effects on the solution. The boundary conditions applied to the system were:

Table 5-2- Summary of Boundary Conditions – Airlift Pump System Mesh Independence Study

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Units</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Inlet &amp; Radial Inlet</td>
<td>Air Mass Flow Inlet</td>
<td>kg/h</td>
<td>1.47 – 13.97</td>
</tr>
<tr>
<td>Suction Inlet</td>
<td>Pressure Inlet</td>
<td>Pa</td>
<td>14,196</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Outlet</td>
<td>Pa</td>
<td>0</td>
</tr>
</tbody>
</table>
Four air mass flow rates were used to generate the performance curve of the airlift pump system. The air flow rates were selected based on the experimental data collected, and were spread out to capture multiple parts of the performance curve.

![Performance Curve Graph](image)

**Figure 5-2: Mesh Independence Performance Results**

Figure 5-2 shows the performance curves obtained from each model. The performance results are quite similar regardless of the element count used. This indicates that a smaller number of elements could be used to obtain a performance curve solution that is mesh independent. Since only the performance curve was recorded from the simulations, a comparison of the velocity or void fraction between the cases was not performed. As mentioned in Chapter 4, the mesh used in the simulation contained 341,243 elements. While the performance curve solution may be mesh independent over the range of the number of elements used, the void fraction tracking used by the VOF model provided better qualitative results with higher mesh densities.
5.2 Single-Phase Flow in The Dual Injection Airlift

The dual injection airlift was at the core of all the work performed in this study. The air injectors essentially dictated the type of flow experienced in the entire airlift pump system, which directly affected the water flow rate out of the pump. This in turn changed the flow in the raceway. This makes understanding the hydrodynamics in the injection system a very important task to reliably model the more complex systems that are an extension of the injectors. Studying the flow in the domain can also provide an insight on how to improve the design of the injection geometry. Two sets of simulations were performed, using one operating fluid at a time for each set. The first set used only air and the second set was only water. The properties of the fluids used in the simulation work is as presented:

Table 5-3- Fluid Properties Used in the Work

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Density</th>
<th>Viscosity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>998.2 $kg/m^3$</td>
<td>1.003x10^{-3} $kg/m/s$</td>
</tr>
<tr>
<td>Air</td>
<td>1.225 $kg/m^3$</td>
<td>1.7894x10^{-5} $kg/m/s$</td>
</tr>
</tbody>
</table>

The same volumetric flow rates were applied to both the air and water cases, thus facilitating comparisons between the two fluid cases. In CFD the inlet boundary conditions were converted to mass flow rates, while considering the density difference of the two fluids. Table 5-4 shows the boundary conditions applied to the model in terms of volumetric rate.
Table 5-4- Summary of Boundary Conditions- Dual Injection Airlift Pump

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Units</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Inlet &amp; Radial Inlet</td>
<td>Volumetric Flow Inlet</td>
<td>$\frac{m^3}{h}$</td>
<td>6 - 60</td>
</tr>
<tr>
<td>Suction Inlet</td>
<td>Pressure Inlet</td>
<td>Pa</td>
<td>0</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Outlet</td>
<td>Pa</td>
<td>0</td>
</tr>
</tbody>
</table>

Understanding the flow in the injectors requires studying multiple flow parameters in the domain to explain the behavior experienced. The parameters of interest are: velocity, vorticity, turbulent kinetic energy, turbulence intensity and pressure. These parameters were monitored at multiple locations in the model as seen in Table 5-5.
Table 5-5- Summary of Solution Parameters Monitored – Dual Injector Airlift Pump

<table>
<thead>
<tr>
<th>Location</th>
<th>Parameters Monitored</th>
</tr>
</thead>
<tbody>
<tr>
<td>① Pressure Inlet</td>
<td>Static Pressure</td>
</tr>
<tr>
<td></td>
<td>Pressure outlet</td>
</tr>
<tr>
<td>① Axial Mass Inlet</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>② Radial Mass Inlet</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>③ Line @ 0.145 m</td>
<td>Velocity Magnitude</td>
</tr>
<tr>
<td>④ Symmetry 1 Boundary</td>
<td>Parameter Contour Plots</td>
</tr>
</tbody>
</table>

This setup provided multiple ways to analyze and compare the solution data, while also facilitating the validation of the numerical results with the experimental results. Since the same boundary conditions were applied to both the air and water cases, the same locations were also used to monitor the parameters. A steady state simulation was
performed for each fluid case, where both the standard K-ε and K-ω turbulence models were tested to find which model provided better predictions.

The following sections discuss the results of each parameter analyzed for a sample case. The inlet boundary condition chosen in the sample case was Q = 27.01 m³/h for both air and water separately. The results presented in this case are from the K-ε turbulence model simulation, and the reasoning is presented in a later section of this work.

5.2.1 Velocity Magnitude

The velocity field was obtained for the flow domain and the results are present in Figure 5-3 where (a) is for the air system and (b) is for the water system.
The velocity distribution is similar throughout the system, which is expected since the same volumetric flow rate is used for both fluids. The velocity of air and water going through the inlet is the same which also matches what is expected by applying:

\[ V = \frac{Q}{A} \]  

(45)

It is also apparent that the fluid velocity increases in the axial injector as it enters the restriction at the manifold. An increase in velocity is also noticed in the radial injector as the flow passes through the perforated section. The difference in velocity between the two fluids is attributed to the difference in density between air and water, since water is heavier and so the velocity that it reached was slightly lower. An interesting observation was made by looking at the orthogonal view of the model, where the effect of each injector
on the flow in the center of the airlift pump can be seen. The axial injector pushes the fluid along the wall of the pipe, while the radial injector pushes the fluid towards the center of the pipe. A radial velocity profile was also obtained at location 3, which is a line located on the symmetry boundary and is 0.145 m high. Figure 5-4 shows the radial velocity profile for both liquids at various inlet conditions.
Figure 5-4 - Air Vs. Water Radial Velocity Profile
5.2.2 Vorticity

Vorticity is a pseudo variable that is derived from the velocity vector field, which describes the local spinning motion of the fluid in a domain. This parameter can indicate the areas where the flow experiences recirculation and eddies. Knowing the location and magnitude of vorticity could indicate where fluid losses are occurring in the domain.
It is not surprising to see that the highest vorticity regions for both fluids are right at the axial manifold and at the radial perforated section. The high vorticity value in the axial manifold can be explained due to the sharp change in geometry coupled with the high velocity magnitude experienced in the region. The increase in vorticity at the radial injection is also due to the restrictive geometry presented at the injector. Vorticity also increases at the walls due to recirculating flows that occur at the boundaries. It is important to note that the vorticity magnitude experienced in the water flow is smaller than that in the air flow. The difference in mass between the two fluids is the main cause of the differences in vorticity magnitude. The vorticity values reached in this model suggest that the flow inside the injectors is very rotational and turbulent.
5.2.3 Turbulent Kinetic Energy

Turbulent kinetic energy is a parameter that describes the mean kinetic energy per mass associated with eddies in the flow. The parameter is obtained though the root mean square of the velocity fluctuations in the domain. The significance of this parameter is knowing where turbulence energy is intensifying, which indicates the energy losses that the system is experiencing.

(a) Air
The turbulent kinetic energy concentration is nearly localized in the axial chamber and injector, with an energy increase present in the radial injector. The kinetic energy always increases when the flow has a high velocity and must go through changes in geometry. It also increases when it is near wall boundaries where the energy becomes concentrated before converting to pressure energy at the wall due to stagnation pressure. These interactions are present in both injectors but the effect is more profound in the axial injector. The kinetic energy experienced by both fluids is also very similar, but since there were small differences in the velocity magnitude, this translated into small differences in the kinetic turbulent energy. Looking at the kinetic energy profiles in the isometric view can identify which areas in the design could be altered to reduce the energy buildup.
5.2.4 Turbulence Intensity

Turbulence intensity is a method to represent the turbulence level as a percentage level. It can be thought of as the fluctuation of flow velocity in comparison to the flow velocity average. Mathematically it is represented as:

\[
I \equiv \frac{u_{rms}}{\bar{u}} \quad (46)
\]

\[
u_{rms} \equiv \sqrt{\frac{1}{3} (u_{rms,x}^2 + u_{rms,y}^2 + u_{rms,z}^2)} = \sqrt{\frac{2}{3}} k \quad (47)
\]

This parameter along with the vorticity describe the rotational nature of the flow in the domain. Physically, turbulence intensity represents the areas in the flow where a lot of mixing occurs. In an airlift pump system, this is could be very beneficial for all heat and mass transfer applications.
Figure 5-7 - Turbulence Intensity Contour Plot: (a) Air, (b) Water
Turbulence intensity in the domain is very high for both fluids. This suggests that the flow constantly mixing and recirculating, especially near the axial air chamber and in the main pipe section right after the radial injector. Studying the intensity values and their relationship with the turbulence kinetic energy and vorticity parameters, indicate that the flow in the injectors is underdeveloped and remains in the inviscid irrotational region in from the inlet and through the injectors.

5.2.5 Static Pressure

Static pressure is also an important flow parameter that describes the hydrodynamics of the flow inside the airlift injectors. Static pressure represents the force that is applied on a certain surface. It also dictates the flow of the fluid in any domain, as a fluid would always move from an area of high pressure to low pressure. Understanding the pressure distribution can also identify the areas causing high energy losses.
The highest-pressure regions are found in the axial chamber. The flow enters from the inlet at a high velocity thus containing high kinetic energy, but as the flow reaches the constriction at the axial manifold, the wall boundary causes a high-pressure region by the manifold. Similarly, looking at the radial injector, the higher-pressure areas are found by the wall boundaries between each radial opening. It can be seen that there is a big difference in the pressure magnitude between the two fluid cases. The mass difference between air and water is the biggest contributor for the pressure magnitude difference. The distribution of pressure in the dual injectors is similar for both cases, which meets the expectations based on the boundary conditions applied.

The static pressure solution from the air flow case was used as the validation parameter of this numerical work. A lab experiment was performed, where the pressure across each of the injector inlets was measured using the flow control meter mentioned in Chapter 3.
and compared to the pressure obtained numerically. The pressure across the injector was calculated by finding the area average of static pressure at the injector inlets and subtracting the area averaged static pressure at the pump pressure outlet. The equations below show the boundaries used to calculate the numerical pressure drop based on the boundary definitions used in Table 5-5.

Axial Pressure Drop: \[ \Delta P = P_1 - P_0 \] (48)
Radial Pressure Drop: \[ \Delta P = P_2 - P_0 \] (49)

It was discussed earlier that both K-\(\varepsilon\) and K-\(\omega\) turbulence models would be tested to compare the results of some of the parameters obtained by each model. Since static pressure is used for validation, it is also an appropriate parameter to which to compare the results of the turbulence models. A pressure drop curve was generated for all the volumetric flow rates used as inlet boundaries.

![Figure 5-9: Dual Injector Air Phase Pressure Drop](image-url)
Figures 5-9 and 5-10 show that although the results are close, it is apparent that the K-\(\varepsilon\) model was able to provide better predictions than the K-\(\omega\) model. The pressure drop solution acquired numerically resulted in a very good agreement with the experimental data, as all the points fall within the error margins of the experimental results. This validation provides confidence in the numerical values obtained from the single-phase simulations.

A static pressure curve for the water flow case was also generated, but no experimental work was done to compare the values obtained numerically. The curve was used to compare the solutions from the K-\(\varepsilon\) and K-\(\omega\) turbulence models in a water only flow case.
The water flow results between the two turbulence models are almost identical. It is therefore not pertinent which model is used for an all water case. Since one of the main purposes of this work is to study two-phase flow in an airlift pump system, the turbulence model selection is an important decision. By looking at the results of the models for each case individually, an assumption can be that the K-ε turbulence model would be more suited for two-phase simulations. The K-ε model was capable of providing better predictions for air flows, while water flow solutions were identical; thus, for an air-water flow simulation, the K-ε turbulence model will be used.
5.3 Two-Phase Flow in an Airlift Pump System

The expansion from the single-phase simulations to two-phase is accompanied by the addition of the upriser pipe to the dual injection airlift to complete the pumping system. The purpose of this simulation is to model the performance of the airlift pump system, while also analyzing the two-phase flow patterns and flow parameters in the upriser pipe. The simulation performed is for an airlift pump system that is operating at a 90% submergence ratio, with various inlet conditions applied. The boundary conditions in this simulation are slightly altered from what was used in the single-phase simulation to accommodate the second phase. The injector inlets (axial & radial) are designated as air flow inlets, while the suction inlet is a water only pressure inlet. Since there is an added upriser pipe attached to the airlift injector that is filled to a certain level with water, a hydrodynamic static pressure is applied at the suction inlet which is equal to the static pressure of the water column at the boundary (density of water by the height of the water column and gravitational acceleration). The summary of the boundary conditions applied to the model are presented in Table 5-6.

Table 5-6: Summary of Boundary Conditions – Airlift Pump System

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Units</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Inlet &amp; Radial Inlet</td>
<td>Air Mass Flow Inlet</td>
<td>$kg/h$</td>
<td>0.22 – 9.56</td>
</tr>
<tr>
<td>Suction Inlet</td>
<td>Pressure Inlet</td>
<td>Pa</td>
<td>14,196</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Outlet</td>
<td>Pa</td>
<td>0</td>
</tr>
</tbody>
</table>

Now that the domain is larger, monitor surfaces were placed at various places in the model to study a variety of parameters, as can be seen in Table 5-7.
Table 5-7: Summary of Solution Parameters Monitored – Airlift Pump System

<table>
<thead>
<tr>
<th>Location</th>
<th>Parameters Monitored</th>
</tr>
</thead>
<tbody>
<tr>
<td>①</td>
<td>Pressure Outlet</td>
</tr>
<tr>
<td>①</td>
<td>Axial Mass Inlet</td>
</tr>
<tr>
<td>②</td>
<td>Radial Mass Inlet</td>
</tr>
<tr>
<td>③</td>
<td>Suction Inlet</td>
</tr>
<tr>
<td>④</td>
<td>Symmetry 1 Boundary</td>
</tr>
<tr>
<td>⑤</td>
<td>Line @ ( Z/d = 5, 10, 15, 20, 25, 30 )</td>
</tr>
</tbody>
</table>
As discussed previously, two-phase flow is a transient phenomenon and as such the solution method used for this case had to be a transient formulation. This is a computationally expensive simulation to run from a stagnant initial condition. The steady state solution was used to provide an initial solution for the transient case. After a certain number of iterations were completed, a switch to the transient formulation was used to record and obtain the final solutions. The transient parameters used for this work included a time step size of $1 \times 10^{-4}$ with 100 iterations running per time step. This method provided a stable system that was capable of reaching residuals smaller than $10^{-3}$ to satisfy the set convergence parameters. The downside of using this method is its inability to capture the flow structure at the start of each new flow case. Laboratory experiments were conducted to find the experimental performance curve of the airlift pump system. As well, the capacitance sensor was used along with the high-speed camera to capture the flow occurring in the upriser pipe. Since a transient solution was obtained for each flow, the various results are presented as time averages for each flow rate, along with an in-depth analysis of each parameter for a sample case where: Air mass flow rate $\dot{m}_{\text{air}} = 2.57 \frac{kg}{h}$, for a period of time $\Delta t = 2.5 \text{ sec}$.

### 5.3.1 Void Fraction

The void fraction portrays the flow structure and the interaction between the two phases in the system. In this simulation $\alpha = 1$ is an all air phase and inversely $\alpha = 0$ is an all water phase. The range between the two values indicates a mixture flow where the value indicates the ratio present from each phase. The void fraction data is used primarily as a qualitative parameter to help visualize the air-water flow in a two-phase domain, as well as to help in identifying the flow pattern experienced in the flow domain. Experimentally, the local void fraction values are found using a capacitance sensor, while a high-speed camera is used to help identify the flow patterns occurring in the upriser pipe. A qualitative comparison between the void fraction obtained by the simulation and the high-speed images obtained from the experimental setup was performed for the conditions at the air flow rate: $\dot{m}_{\text{air}} = 2.57 \frac{kg}{h}$ at $\frac{Z}{d} = 15$ over a period of time $\Delta t = 2.5 \text{ sec}$. The results of the
comparison can indicate whether the flow patterns experienced in the simulation domain and the physical flow are similar.
Figure 5-12: Void Fraction Time Sequence at $\frac{Z}{d} = 15$: (a) Numerical Simulation (b) Experimental
Comparing the results obtained, it can be observed that both flows are experiencing slug flow. Slug flow is characterized by large air bubbles with diameters close to the pipe diameter. It can also be seen that there is a constant presence of smaller bubbles or mixture flows in the domain. This is indicated by the void fraction values in the range of $50 < \alpha < 90$. A point of interest that can be seen both in the physical system and the simulation is the tail of an air slug. After an air slug passes through the upriser tube, it is followed by smaller bubbles in close proximity to the bottom of the slug. This is a complex flow phenomenon and it shows the interphase tracking capabilities of the VOF model. Looking at the void fraction distribution along the entire airlift pump system, the evolution of the air bubble movements can be observed.
Figure 5-13- Void Fraction in Airlift Pump System Time Sequence
It is of interest to track the movement as well as the coalescence and breakup of the bubbles as they travel upwards in the upriser pipe. The effect of the slug bubbles can also be seen as they act as air pistons carrying the water volume above them. A radial profile of the void fraction was also captured by evaluating the time averaged value for the same case. This shows the average distribution of each phase across the radius of the upriser pipe at various distances. The radial distribution can show the effects of each of the injectors in the dual injection airlift pump.
Figure 5-14- Time Averaged Radial Void Fraction Profile at $m_{air} = 2.57 \frac{kg}{h}$
Looking at $\frac{Z}{d} = 5$, which is the closest distance to the airlift injectors, it is observed that the air is concentrated near the wall where the effects of the axial injectors are more present than those of the radial injector. Moving upwards along the upriser pipe, the void fraction becomes more evenly distributed near the center of the pipe. The results show that the water phase is more present at each level of the upriser pipe. This is reasonable since with a 90% submergence level system and time averaged values, water should be the most dominant phase. An area weighted average was also found for the void fraction at each airflow rate in the upriser pipe. This would indicate the overall trend of the phase distribution experienced in the upriser pipe with the change of the amount of air injected into the system.
Figure 5.15- Void Fraction Vs. Air Mass Flow Rate
The trend in Figure 5-15 shows the change in void fraction averages for each air flow rate. The results show that the highest void fraction percentage occurs at $\frac{Z}{d} = 5$ and then drops slightly, moving upwards in the upriser pipe where it remains quite consistent. This is also very reasonable and matches what is expected in this flow.

### 5.3.2 Velocity

The flow velocity was also computed and studied in the airlift pump system. The velocities presented in this section are the mixture velocities of the two-phases. With the knowledge obtained from studying the void fraction distribution from the previous section, it could be estimated what is the velocity of each phase in the domain. Velocity is a very important parameter to analyze because of its effect on the performance of an airlift pump system. The velocity magnitude along with its directional vector show the movement of the flow in the upriser pipe. Understanding the velocity distribution in the domain can also provide an understanding of the effects of the injectors on the two-phase flow in the upriser pipe. A phenomenon such as reverse flow is also detected by studying the velocity in the domain. The radial distribution of the time averaged mixture velocity was found at the various heights of the upriser pipe.
Figure 5-16 - Time Average Radial Velocity Profile at $\dot{m}_{\text{air}} = 2.57 \frac{kg}{\text{s}}$
The velocity distribution shows how the velocity is affected throughout the pipe. It can be observed that at \( \frac{Z}{d} = 5 \) which is nearest to the injectors, the velocity close to the axial injector is higher than that in the rest of the domain. This trend was similarly observed in the air and water single phase flow simulations covered previously. This shows the purpose of the axial injector in the dual injector airlift pump, where it pushes air at higher velocities in the domain to aid the wall flows along the domain. The velocity distribution becomes more evenly distributed along the center of the pipe. A shift in velocity occurs similar to that of the void fraction distribution. This is because the air bubbles have higher velocities than that of the water around them, so as the bubbles start moving towards the center of the pipe, the mixture velocity distribution changes in the same direction. Figure 5-17 presents the velocity contour plot at the airlift injector, while Figure 5-18 presents the velocity vector plot at \( \frac{Z}{d} = 15 \).
Figure 5-17 - Velocity Contour at the Airlift Injector
<table>
<thead>
<tr>
<th>t = 1 sec</th>
<th>1.1 sec</th>
<th>1.2 sec</th>
<th>1.3 sec</th>
<th>1.4 sec</th>
</tr>
</thead>
</table>

Figure 5-18 - Velocity Vector at $\frac{z}{d} = 15$ and Airlift Injector
The velocity vectors show the direction of the flow in the various sections of the domain. Looking at the vector field for $\frac{Z}{d} = 15$ at the time sequence from 1 to 1.4 seconds, it can be seen that some reverse flow occurs at the side of the pipe. This time snippet coincides with the passing of the slug through that section, which causes some of the flow to fall downwards at the wall. The area weighted time average of the velocity magnitude was also computed for the various air flow rates used in this study and presented in Figure 5-19. This provides an overall understanding of the velocity magnitude when there is a change in the air flow at the inlet.
Figure 5-19- Area Weighted, Time Average Velocity Magnitude Vs. Air Flow Rate
The results show that with an increase in the amount of air injected, the velocity increases as well. This trend matches what would be expected for the velocity profiles. Since the values are averaged over the flow time of each case, the velocity magnitudes are consistent throughout the entirety of the upriser domain.

5.3.3 Pressure

The static pressure field was computed throughout the upriser pipe to study and analyze the effects that the flow might have on the pressure field. An area weighted and time averaged plot of the static pressure was captured at the various heights of the upriser pipe and presented in Figure 5-20.

![Figure 5-20- Area Weighted, Time Averaged Static Pressure Vs. Air Flow Rate](image)
The pressure profile obtained indicates that the static pressure in the upriser pipe remains fairly constant regardless of the air flow conditions applied to system. This trend is in agreement with the experimental trend found by Moisidis and Kastrinakis [147]. This leads to the conclusion that during the operation of the airlift pump, the pressure field is not affected by the flow experienced in the domain.

5.3.4 Performance Curve

The performance curve describes the output of the system. This is the amount of water pumped in relation to the input, which is the amount of air injected into the airlift pump. This identifies the operational range of the pump based on the application needed. Obtaining the performance curve is essential for the design process of any airlift pump system. Using CFD to model the performance is a great step forward in two-phase simulation work and the design of multi-phase systems. The performance was generated using the time averaged water flow rate that was recorded at the outlet of the airlift pump system using various inlet air flow rates. An experimental performance curve was also generated using the laboratory setup to validate the results of the simulation. In addition to the experimental and numerical curves, a performance curve was also generated using an analytical model developed by Kassab et al. [36]. This model was created based on the one-dimensional analysis of a separated two-phase flow model, where the inputs to the model are the size of the airlift pump, the length of the pipe and the submergence ratio to find the performance curve. The limitation of the analytical model is due to the one-dimensional formulation which does not consider the injector geometry or their effect on the flow. The model also assumed slug flow as the only flow pattern experienced which is not what was found experimentally or through numerical simulation. Comparison between the models can indicate how well the prediction methods agree with each other and the experimental data. Figure 5-21 shows the performance curves of the airlift pump obtained experimentally, analytically and numerically. Figure 5-22 compares the predictive models against the measured values obtained.
Figure 5-21 - Airlift Pump Performance Curve

The diagram shows the relationship between air flow rate (kg/h) and water flow rate (kg/h) for an airlift pump. The experimental data is represented by black squares, while the CFD modeling results are shown in blue circles. The black line represents the experimental trend line, and the blue dotted line represents the CFD trend line. The coefficient of determination, $R^2 = 0.9825$, indicates a strong correlation between the variables.
Figure 5-22 - Measured Vs. Predicted Water Flow Rate
Looking at Figure 5-21, the performance of an airlift pump initially increases with an increase in the air flow rate injected into the system. It reaches a point of maximum flow, and then starts to decrease gradually with each further increase in the amount of air injected into the system. This trend is visible in all the performance curves obtained experimentally, numerically, and analytically. The results obtained numerically through the CFD simulation appear to have very good agreement with the experimental results and fall very close within the measurement uncertainty value which is at ±5%. It appears that the numerical solutions are consistently over predicting the outlet water flow rate. The difference between the numerical and the experimental values increased after the maximum water flow point. This is due to the change in the flow pattern in the upriser pipe, which becomes more complex, making it difficult to model. A trend line was generated based on the numerical results with a correlation value of 98.25%. The trend line can help in providing an estimate of the airlift pump performance in between the data points. From Figure 5-22, the predictive capabilities of the CFD model are within a 20% margin of error, which outperforms the analytical model presented. These results prove that computational fluid dynamics can be used as a valid predictive method for modelling the operation of airlift pumps.

As mentioned in Chapter 2, the performance of an airlift pump is dependent on the flow pattern experienced in the domain. The flow pattern experienced is typically determined qualitatively by visually inspecting the flow structure in the upriser pipe. There have also been analytical and theoretical methods developed to determine the expected flow pattern experienced in a domain through the use of flow pattern maps. A flow pattern map attempts to represent the flow structure by tying it to a flow parameter, typically a certain component of the flow rates chosen for the modelling. These maps vary depending on the orientation of the flow, the geometry of the domain and many other parameters. A flow pattern map developed by Taitel et al. [64] matches the flow conditions found in this work so it will be used to find and confirm the flow patterns occurring in the domain. This map is based on the superficial velocities of each phase in the domain. Superficial velocity is an artificial flow velocity that is defined by the volumetric flow rate of a phase over the
total cross-sectional area of the flow domain. The equation assumes that only a single phase is present in the domain where it is numerically presented as:

\[ j_g = \frac{Q_g}{A} \]  \hspace{1cm} (50)

\[ j_l = \frac{Q_l}{A} \]  \hspace{1cm} (51)

The superficial velocities for each phase were calculated for each flow case to determine their location on the flow pattern map. The flow pattern map is presented in the Figure 5-23.
The flow pattern map identified the flow pattern for the CFD simulation and the experimental flow. The results show that most flow patterns which occur in the domain are slug flow or on the transition line between bubbly and slug flow. The similarities between the experimental and the numerical results are due to the similarity in the water flow rate between the two solutions. This does not guarantee that the flow pattern experienced in both cases is exactly the same, as the visual identification of the flow pattern occurring remains the best way of characterizing the flow structure. An observation at each set of flow parameters was performed to indicate the flow pattern experienced. Comparing the flow patterns experienced as seen in Figure 5-23 shows that experimentally the flow pattern changes occur sooner than what the flow pattern map
predicts. While the computational results match the flow pattern map prediction than what was found experimentally. The numerical simulation of highly complex and turbulent flow patterns is difficult, such that the VOF model attempts at remaining in a flow pattern where tracking the interface is more achievable. This comparison provides a better understanding of the interactions that occur in a two-phase flow system, while remaining entirely for qualitative purposes.
5.4 Flow in an Aquaculture Raceway

The raceway simulation was the culmination of all the previous work. The airlift pump system was tested on an aquaculture raceway as a study case for the operation of a dual injection airlift pump. This could be used as a design case for future projects. The ability to test various airlift system designs, and determine their performance impact on any system would be a huge advantage in the commercialization process of aquaculture facilities. Based on all the work presented for the $2.54 \times 10^{-2}$ m diameter airlift pump, it was established that the CFD model could produce accurate predictions with a very good agreement with the experimental data. This confidence permitted the assumption that the model could provide a good prediction for the $1.016 \times 10^{-1}$ m diameter airlift pump used in the raceway. The raceway setup mentioned in Chapter 4 was a section of an existing raceway in an aquaculture facility. The transient parameters used for this work included a time step size of $1 \times 10^{-2}$ with 40 iterations running per time step. Water velocity measurements were taken at different locations to provide a velocity profile to which the simulations were compared. The boundary conditions applied to the raceway model are presented in Table 5-8.

### Table 5-8-Summary of Boundary Conditions - Raceway

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Units</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water Inlet</td>
<td>Mass Flow Inlet</td>
<td>$kg \over h$</td>
<td>Water: 27,000</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Air: 22.68</td>
</tr>
<tr>
<td>Water Outlet</td>
<td>Outflow</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Air Outlet</td>
<td>Outflow</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
The velocity magnitudes were obtained from several locations from the model as seen in Table 5-9.

**Table 5-9- Summary of Solution Parameters Monitored - Raceway**

<table>
<thead>
<tr>
<th>Location</th>
<th>Parameters Monitored</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points along lengths:</td>
<td>Velocity Magnitude</td>
</tr>
<tr>
<td>0.93, 1.22, 2.03, 2.74, 3.35, 4.06 m</td>
<td></td>
</tr>
<tr>
<td>Z = 0.3, 0.35, 0.43 m</td>
<td>X-Direction Velocity Component</td>
</tr>
</tbody>
</table>

The x-component of the velocity was measured in the raceway experimentally using the acoustic doppler velocimeter. Velocity monitors were placed at a total of 48 points at various lengths and depths in the raceway. Matching the experimental measurement plan outlined in Chapter 3, velocity monitors were placed along the center line of the raceway as well on the symmetry boundaries. The rest of the points were placed on the symmetry boundaries to capture the velocity magnitude at the intersection between the other sections of the raceway.
Figure 5-25: X-Direction Velocity Profile in Raceway
The velocity comparison between the numerical solution and the experimental measurements show that the majority of the data points fall within the experimental error bars provided from the ADV. Some deviation between the two results also exists. This is apparent at $Z = 0.3m$ which is the shallowest depth used to collect any measurements. At this depth, the flow was turbulent due to water recirculation near the surface of the raceway. This increased the percentage error in the velocity values obtained by the velocimeter. The experimental velocity magnitudes at one of the symmetry boundaries also seemed to have different values when compared to the numerical solution, or even the other symmetry boundary. This difference was present because it was found that the raceway and the airlift system installed at the facility were slightly asymmetrical. This caused a concentration in the flow to occur more to one side of the raceway than the other. Those velocity readings still show the overall velocity magnitudes required for the operation of the raceway despite the slight bias caused by the construction of the system. The numerical solutions obtained through the simulation show that the same velocity magnitudes were achieved through computational fluid dynamics. Utilizing this method, it would be possible to use CFD to predict the operational flows in future designs of aquaculture systems while still being in the design phase of a project. This could lead to further innovative designs and their implementation into aquaculture systems, helping to make such systems more economical and widespread.
Chapter 6 - Conclusion

Numerical simulations were performed on multiple components of an airlift pump system integrated into an aquaculture application. The dual injection airlift model was analyzed to study the hydrodynamics of the flow and the effects of each injector. It was found that the axial injector provided higher flow velocities at the wall, while the effects of the radial injectors were more profound near the center of the airlift pump. The pressure drop obtained across the airlift pump using a numerical simulation was validated against experimental pressure readings at the injectors. Based on the validation it was determined that the K-ε turbulence model would be more appropriate for further simulations. Once the single-phase flow characteristics in the airlift pump were obtained and analyzed, a two-phase numerical simulation was performed on a complete airlift pump system.

The simulation was performed to model the performance of the airlift pump, as well as to study the two-phase phenomenon in the upriser pipe. It was found that the numerical simulation of the airlift pump system provided accurate predictions on the performance parameters of the pump. Some two-phase flow patterns were also observed through the numerical simulation and a comparison between the experimental and numerical results was made using a flow pattern map. The validity of the numerical results was established when a comparison between the numerical, experimental and analytical studies of the water outlet solution was performed.

Predictions resulting from the numerical solution were within a 20% margin of error and outperformed the predictions of the analytical model. This validation permitted the current study to integrate a larger diameter airlift pump into an aquaculture raceway system. This research was performed to measure the velocity profiles along the raceway and discover if the pump would be able to provide the required velocities for operation. A section of the raceway was modelled and a free surface simulation was performed to capture the interaction between the air-water inlet of the airlift with the raceway boundary.
The results showed that when using CFD, the predicted velocity magnitude matched that which was observed in the field. These results showed that the use of computational fluid dynamics could be integrated into the design process of airlift pump systems. This would permit the integration of these pumps into aquaculture systems by obtaining an accurate prediction of the operating conditions. This could have a massive economic potential, as a numerical study could be performed before having to build the physical facilities. This would result in optimized solutions that would make airlift pumps and aquaculture more economically attractive and thus help in the growth of the industry.
REFERENCES


[12] A. Armyagov, “Farm salmon fishing in Norway aerial photography.”.


[16] M. P. Masser, “Cage Culture in Freshwater and Protected Marine Areas,” in


[27] Pranger, “Indoor circular tank.”


dynamics (CFD) for aquaculture raceway design to increase settling

hydrodynamics and path/residence time of aquaculture-like particles in a mixed-
cell raceway (MCR) using 3D computational fluid dynamics (CFD),” Aquac. Eng.,

simulation of the hydrodynamics within octagonal tanks in recirculating

energy efficiency of different high rate algal raceway pond designs using
226, 2013.


[137] L. Euler, “Principes généraux de l’état d’équilibre des fluides; Principes généraux
du mouvement des fluides; Continuation des recherches sur la théorie du
mouvement des fluides,” Hist. l’Académie Berlin, 1755.


[141] O. Ubbink, “Numerical prediction of two fluid systems with sharp interfaces,”

123–164, 1895.

[143] B. E. (Brian E. Launder and D. B. Spalding, Lectures in mathematical models of


APPENDIX A: EXPERIMENTAL DATA

Table 6-1 - Experimental Pressure Measurements - Air

<table>
<thead>
<tr>
<th>Experimental</th>
<th>Air Volume Flow Rate (m^3/h)</th>
<th>Axial Pressure (pa)</th>
<th>Radial Pressure (pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>6.14</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>15.19</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>20.94</td>
<td>68.94</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>27.01</td>
<td>137.89</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>39.88</td>
<td>413.68</td>
<td>68.94</td>
</tr>
<tr>
<td></td>
<td>46.62</td>
<td>620.52</td>
<td>137.89</td>
</tr>
<tr>
<td></td>
<td>53.30</td>
<td>896.31</td>
<td>137.89</td>
</tr>
<tr>
<td></td>
<td>60.15</td>
<td>1310.00</td>
<td>206.84</td>
</tr>
</tbody>
</table>

Table 6-2 - Experimental Data for 2.54 Airlift Pump System

<table>
<thead>
<tr>
<th>Experimental</th>
<th>Air Mass Flow (kg/h)</th>
<th>Total Water Mass Flow (kg/h)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.22</td>
<td>833.29</td>
</tr>
<tr>
<td></td>
<td>0.74</td>
<td>1632.50</td>
</tr>
<tr>
<td></td>
<td>1.47</td>
<td>2050.97</td>
</tr>
<tr>
<td></td>
<td>1.84</td>
<td>2308.28</td>
</tr>
<tr>
<td></td>
<td>2.21</td>
<td>2343.30</td>
</tr>
<tr>
<td></td>
<td>2.57</td>
<td>2410.68</td>
</tr>
<tr>
<td></td>
<td>2.94</td>
<td>2371.03</td>
</tr>
<tr>
<td></td>
<td>3.68</td>
<td>2280.55</td>
</tr>
<tr>
<td></td>
<td>4.41</td>
<td>2164.99</td>
</tr>
<tr>
<td></td>
<td>5.88</td>
<td>1934.63</td>
</tr>
<tr>
<td></td>
<td>9.56</td>
<td>1544.93</td>
</tr>
</tbody>
</table>
### Table 6-3: Experimental Raceway Velocities

<table>
<thead>
<tr>
<th>Length</th>
<th>Symmetry Velocity (m/s)</th>
<th>Pump Velocity (m/s)</th>
<th>Symmetry Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.93</td>
<td>*</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td>1.22</td>
<td>0.044</td>
<td>0.153</td>
<td>0.039</td>
</tr>
<tr>
<td>2.03</td>
<td>0.051</td>
<td>0.052</td>
<td>0.203</td>
</tr>
<tr>
<td>2.74</td>
<td>0.079</td>
<td>0.075</td>
<td>0.234</td>
</tr>
<tr>
<td>3.35</td>
<td>0.057</td>
<td>0.052</td>
<td>0.165</td>
</tr>
<tr>
<td>4.06</td>
<td>*</td>
<td>*</td>
<td>*</td>
</tr>
</tbody>
</table>

**Z = 0.35**

<table>
<thead>
<tr>
<th>Length</th>
<th>Symmetry Velocity (m/s)</th>
<th>Pump Velocity (m/s)</th>
<th>Symmetry Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.93</td>
<td>0.129</td>
<td>0.149</td>
<td>0.073</td>
</tr>
<tr>
<td>1.22</td>
<td>0.136</td>
<td>0.165</td>
<td>0.034</td>
</tr>
<tr>
<td>2.03</td>
<td>0.074</td>
<td>0.128</td>
<td>0.136</td>
</tr>
<tr>
<td>2.74</td>
<td>0.070</td>
<td>0.046</td>
<td>0.185</td>
</tr>
<tr>
<td>3.35</td>
<td>0.048</td>
<td>0.073</td>
<td>0.157</td>
</tr>
<tr>
<td>4.06</td>
<td>0.029</td>
<td>0.043</td>
<td>0.150</td>
</tr>
</tbody>
</table>

**Z = 0.43**

<table>
<thead>
<tr>
<th>Length</th>
<th>Symmetry Velocity (m/s)</th>
<th>Pump Velocity (m/s)</th>
<th>Symmetry Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.93</td>
<td>0.156</td>
<td>0.188</td>
<td>0.125</td>
</tr>
<tr>
<td>1.22</td>
<td>0.172</td>
<td>0.169</td>
<td>0.132</td>
</tr>
<tr>
<td>2.03</td>
<td>0.148</td>
<td>0.144</td>
<td>0.115</td>
</tr>
<tr>
<td>2.74</td>
<td>0.081</td>
<td>0.067</td>
<td>0.137</td>
</tr>
<tr>
<td>3.35</td>
<td>0.083</td>
<td>0.076</td>
<td>0.107</td>
</tr>
<tr>
<td>4.06</td>
<td>0.021</td>
<td>0.058</td>
<td>0.050</td>
</tr>
</tbody>
</table>
### APPENDIX B: COMPUTATIONAL DATA

#### Table 6-4: Computational Pressure Monitors - Air

<table>
<thead>
<tr>
<th>CFD</th>
<th>K-e Axial Pressure (pa)</th>
<th>K-e Radial Pressure (pa)</th>
<th>K-w Axial Pressure (pa)</th>
<th>K-w Radial Pressure (pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air Volume Flow Rate (m³/h)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.14</td>
<td>7.75</td>
<td>1.24</td>
<td>10.75</td>
<td>1.24</td>
</tr>
<tr>
<td>15.19</td>
<td>40.47</td>
<td>8.08</td>
<td>41.09</td>
<td>8.08</td>
</tr>
<tr>
<td>20.94</td>
<td>79.05</td>
<td>15.94</td>
<td>91.45</td>
<td>15.94</td>
</tr>
<tr>
<td>27.01</td>
<td>141.60</td>
<td>28.33</td>
<td>146.76</td>
<td>28.33</td>
</tr>
<tr>
<td>39.88</td>
<td>410.79</td>
<td>70.89</td>
<td>366.67</td>
<td>70.89</td>
</tr>
<tr>
<td>46.62</td>
<td>628.15</td>
<td>109.72</td>
<td>623.38</td>
<td>109.72</td>
</tr>
<tr>
<td>53.30</td>
<td>957.46</td>
<td>172.44</td>
<td>946.23</td>
<td>172.44</td>
</tr>
<tr>
<td>60.15</td>
<td>1343.76</td>
<td>247.22</td>
<td>1464.60</td>
<td>247.22</td>
</tr>
</tbody>
</table>

#### Table 6-5: Computational Pressure Monitor - Water

<table>
<thead>
<tr>
<th>CFD</th>
<th>K-e Axial Pressure (pa)</th>
<th>K-e Radial Pressure (pa)</th>
<th>K-w Axial Pressure (pa)</th>
<th>K-w Radial Pressure (pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air Volume Flow Rate (m³/h)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.14</td>
<td>4829.70</td>
<td>1006.73</td>
<td>4917.90</td>
<td>980.43</td>
</tr>
<tr>
<td>15.19</td>
<td>29000.05</td>
<td>6150.23</td>
<td>29015.65</td>
<td>6098.36</td>
</tr>
<tr>
<td>20.94</td>
<td>55070.70</td>
<td>12329.20</td>
<td>54834.67</td>
<td>10672.34</td>
</tr>
<tr>
<td>27.01</td>
<td>91376.70</td>
<td>19751.92</td>
<td>90983.84</td>
<td>17482.74</td>
</tr>
<tr>
<td>39.88</td>
<td>198208.27</td>
<td>42700.33</td>
<td>197108.27</td>
<td>40340.35</td>
</tr>
<tr>
<td>46.62</td>
<td>270703.88</td>
<td>58101.51</td>
<td>271315.66</td>
<td>51663.76</td>
</tr>
<tr>
<td>53.30</td>
<td>353269.49</td>
<td>75237.67</td>
<td>354282.99</td>
<td>68218.08</td>
</tr>
<tr>
<td>60.15</td>
<td>449387.02</td>
<td>94729.17</td>
<td>450958.92</td>
<td>85796.41</td>
</tr>
</tbody>
</table>
Table 6-6: Computational Performance Data

<table>
<thead>
<tr>
<th>CFD</th>
<th>Air Mass Flow (kg/h)</th>
<th>Water Mass Flow (kg/h)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.22</td>
<td>855.80</td>
</tr>
<tr>
<td></td>
<td>0.73</td>
<td>1675.71</td>
</tr>
<tr>
<td></td>
<td>1.47</td>
<td>2014.19</td>
</tr>
<tr>
<td></td>
<td>1.84</td>
<td>2395.07</td>
</tr>
<tr>
<td></td>
<td>2.21</td>
<td>2539.44</td>
</tr>
<tr>
<td></td>
<td>2.57</td>
<td>2578.00</td>
</tr>
<tr>
<td></td>
<td>2.94</td>
<td>2491.20</td>
</tr>
<tr>
<td></td>
<td>3.69</td>
<td>2462.40</td>
</tr>
<tr>
<td></td>
<td>4.41</td>
<td>2344.32</td>
</tr>
<tr>
<td></td>
<td>5.88</td>
<td>1892.43</td>
</tr>
<tr>
<td></td>
<td>9.56</td>
<td>1501.53</td>
</tr>
</tbody>
</table>
## Table 6-7: Computational Raceway Velocities

<table>
<thead>
<tr>
<th>Length</th>
<th>Symmetry Velocity (m/s)</th>
<th>Center Line Velocity (m/s)</th>
<th>Symmetry Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.93</td>
<td>*</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td>1.22</td>
<td>0.042</td>
<td>0.176</td>
<td>0.042</td>
</tr>
<tr>
<td>2.03</td>
<td>0.092</td>
<td>0.066</td>
<td>0.092</td>
</tr>
<tr>
<td>2.74</td>
<td>0.096</td>
<td>0.072</td>
<td>0.096</td>
</tr>
<tr>
<td>3.35</td>
<td>0.111</td>
<td>0.076</td>
<td>0.111</td>
</tr>
<tr>
<td>4.06</td>
<td>*</td>
<td>*</td>
<td>*</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Length</th>
<th>Symmetry Velocity (m/s)</th>
<th>Center Line Velocity (m/s)</th>
<th>Symmetry Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.93</td>
<td>0.154</td>
<td>0.086</td>
<td>0.154</td>
</tr>
<tr>
<td>1.22</td>
<td>0.018</td>
<td>0.170</td>
<td>0.018</td>
</tr>
<tr>
<td>2.03</td>
<td>0.062</td>
<td>0.087</td>
<td>0.062</td>
</tr>
<tr>
<td>2.74</td>
<td>0.055</td>
<td>0.033</td>
<td>0.055</td>
</tr>
<tr>
<td>3.35</td>
<td>0.078</td>
<td>0.041</td>
<td>0.078</td>
</tr>
<tr>
<td>4.06</td>
<td>0.065</td>
<td>0.042</td>
<td>0.065</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Length</th>
<th>Symmetry Velocity (m/s)</th>
<th>Center Line Velocity (m/s)</th>
<th>Symmetry Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.93</td>
<td>0.155</td>
<td>0.195</td>
<td>0.155</td>
</tr>
<tr>
<td>1.22</td>
<td>0.181</td>
<td>0.179</td>
<td>0.181</td>
</tr>
<tr>
<td>2.03</td>
<td>0.080</td>
<td>0.084</td>
<td>0.080</td>
</tr>
<tr>
<td>2.74</td>
<td>0.114</td>
<td>0.062</td>
<td>0.114</td>
</tr>
<tr>
<td>3.35</td>
<td>0.034</td>
<td>0.072</td>
<td>0.034</td>
</tr>
<tr>
<td>4.06</td>
<td>0.021</td>
<td>0.057</td>
<td>0.021</td>
</tr>
</tbody>
</table>