NUMERICAL INVESTIGATION ON THE FLOW CHARACTERISTICS OF THREE PHASE FLOW IN MICROCHANNEL

A PROJECT REPORT

Submitted by

ANAND V B	**KKE20CH008**
SUJISHA V C	**KKE20CH054**
VISHNUPRIYA M	**KKE20CH061**
RITHIKA SAMSON	**KKE20CH065**

to

The APJ Abdul Kalam Technological University

in partial fulfillment of the requirements for the award of the Degree

of

Bachelor of Technology

in

Chemical Engineering

Department of Chemical Engineering

Government Engineering College, Kozhikode

MAY 2024

DECLARATION

We hereby declare that the Project report "**NUMERICAL INVESTIGATION ON THE FLOW CHARACTERISTICS OF THREE-PHASE FLOW IN MICROCHANNEL**", submitted for partial fulfillment of the requirements for the award of the degree of Bachelor of Technology of the APJ Abdul Kalam Technological University, Kerala is a bonafide project report done by us under supervision of **Prof. DHANEESH K P**. The submission represents our ideas in our own words and where ideas or words of others have been included, we have adequately and accurately cited and referenced the original sources. We also declare that we have adhered to ethics of academic honesty and integrity and have not misrepresented or fabricated any data or idea or fact or source in our submission. We understand that any violation of the abovewill be a cause for disciplinary action by the institute and/or the University and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been obtained. This report has not been previously formed the basis for the award of any degree, diploma, or similar title of any other University.

Place: Kozhikode

Signature

Date: 23/04/2024

ANAND V B
SUJISHA V C
VISHNUPRIYA M
RITHIKA SAMSON

DEPARTMENT OF CHEMICAL ENGINEERING

GOVERNMENT ENGINEERING COLLEGE, KOZHIKODE

CERTIFICATE

This is to certify that the report entitled **NUMERICAL INVESTIGATION ON THE FLOW CHARACTERISTICS OF THREE-PHASE FLOW IN MICROCHANNEL** submitted by ANAND V.B, SUJISHA V.C, VISHNUPRIYA M, RITHIKA SAMSON to the APJ Abdul Kalam Technological University in partial fulfillment of the requirements for the award of the Degree of Bachelor of Technology in Chemical Engineering is a Bonafide record of the project presented by her. This report in any form has not been submitted to any other University or Institute for any purpose.

PROJECT GUIDE
Prof. DHANEESH K P
Assistant Professor
Department of Chemical Engineering
GEC Kozhikode

HEAD OF THE DEPT
Dr. ANJANA R
Associate Professor
Department of Chemical Engineering
GEC Kozhikode

CONTENTS

Contents Page no.

ACKNOWLEDGEMENT

We had immense pleasure in presenting our project "**NUMERICAL INVESTIGATION ON THE FLOW CHARACTERISTICS OF THREE PHASE IN MICROCHANNEL**", we would like to express our sincere gratitude to all those who have contributed directly or indirectly to make this seminar a success. First of all, we would like to thank the God almighty for his kind blessings upon us for the completion of this seminar.

We are deeply indebted to **Dr. Anjana R**, Head of the Department of Chemical Engineering, Government College of Engineering, Kozhikode and to our guide **Prof. Dhaneesh K P** for his valuable guidance and kind supervision given to us throughout the course which shaped the present work as its show.

We are also grateful to our project co-coordinators **Dr. Manju M S, Dr. Binu T V** and **Prof. Jenny N.**

We surrender our esteemed and sincere thanks to Faculty members of the Chemical Engineering Department for their inspiration and encouragement. Last but not least we thank our parents, friends, and all others who have assisted and supported us for the completion of our work.

ABSTRACT

Our work primarily aims at observing the complex dynamics associated with the three phase microflows in a T-junction microchannel. Various numerical techniques like computational fluid dynamics (CFD) are resorted to analyze the pattern of flow and characteristics along various flow regimes and geometrical setups within the microchannel. The results which are thus obtained are essential for applications in microfluidic systems. The mathematical models are selected accordingly so that the equations associated with the model provides a numerically reliable approach enabling precise depiction. The study dives deep into the extent to which various parameters influence the flow through the microchannel. Our work primarily concentrates on the cross-junction geometry providing novel insights into the associated three phase interactions. This data can be crucial in the design and optimization of microfluidic devices. We aim in deepening our knowledge on intricate three-phase microflows which have implications in diverse fields, including biomedical devices and chemical processing.

Keywords: three-phase flow, CFD, microchannel

LIST OF FIGURES

LIST OF TABLES

ABBREVIATIONS

1. CFD – Computational Fluid Dynamics
2. FOAM – Field Operation and Manipulation
3. ICEM – Integrated Computer Engineering and Manufacturing
4. ID – Identification
5. VOF – Volume of Fluid

CHAPTER 1

INTRODUCTION

1.1 GENERAL BACKGROUND

CFD is a science that helps in quantitively analyzing various flow patterns in fluids with help of computers. The laws on conservation of mass, energy and momentum forms the basic foundation of CFD technique. CFD is a versatile technique which has widespread applications in fluid mechanics and engineering operations. Complex flow patterns and flow dynamics can be modelled using various numerical methods even scaling down to microscopic level. CFD can be resorted to analyze complex flow patterns through microchannel. CFD can also provide additional information which cannot be attained through conventional analysis techniques thereby giving additional dimensions of microflow. CFD also aids in non-invasive flow measurement without providing any disruption to the existing flow.

Microchannels are flow channels which can be made useful in the analysis of complex flow dynamics. It also aids in fine tuning of small three phase microflows. Microfluidic devices are capable of handling high throughput and high time-to-quantity flows which aims at analyzing the complex geometry between the three phases. High mass to mass and heat transfer capabilities of microfluidic systems imparts them with high efficiency especially in the usage in drug delivery systems and microscale process of heat exchangers.

The T-junction geometry, which can split a single stream into two output streams, is used for three-phase flows. Phase maldistribution phenomena is shown to occur unavoidably when a two-phase flow enters the dividing T-junction. Stated differently, the two output streams from the T-junction—one rich in liquid and the other high in vapor—often have differing qualities from one another. A phase separation device may be created from the unequal distribution of T-junction, but it also has the potential to seriously harm downstream facilities. For example, in the petrochemical sector, steam is initially injected to aid in the extraction of petroleum, and it is subsequently separated from oil, with the aid of a T-junction to enable pre-separation.

Based on their structural characteristics, T-junctions can be divided into two categories: branching T-junctions and impacting T-junctions. The stream flows straight forward from one of the two branches at a branching T-junction. The two outflow streams for an impacting type, however, are perpendicular to the entrance stream. Energy systems make extensive use of both categories. In order to prevent inhomogeneous heat transfer and lower heat exchanger performance, a distributor is employed to distribute the two-phase refrigerant mixture evenly over the many parallel pipes. Some tubes may be rich in liquid while others are rich in gas. In several other situations, the T-junction's phase separation capacity can be utilized to accomplish certain goals, like in some lately suggested power generating, heat pump, or refrigeration cycles to enhance energy management and conversation. However, there are several thresholds for improved use in energy systems due to the intricacy of the influence of influencing factors on phase separation.

Microfluidic systems play a pivotal role in various industries, ranging from biomedical applications like drug delivery to engineering applications such as microscale heat exchangers. By precisely analyzing the flow patterns and mixing efficiencies within a cross junction microchannel, our project contributes to the foundational understanding of complex interactions at the microscale. This information is essential for developing and refining microfluidic devices that can function more dependably and efficiently. The project's real-world significance ultimately rests in its ability to open the door for creative and effective microfluidic solutions that tackle problems in a variety of industries.

The exact knowledge obtained about mixing efficiencies, and flow patterns inside microchannels can help designers of drug delivery systems create more efficient ones. Potential uses for optimizing three-phase microflows include raising the effectiveness of microscale heat exchangers. Furthermore, the project findings can advance the design of microfluidic systems for chemical processing, where the understanding of complex interactions between different phases is crucial. Overall, the project holds promise for innovations in microfluidics with broad-reaching impacts across biomedical, engineering, and chemical processing domains.

1.2 OBJECTIVES

The objectives of our project can be outlined as follows:

- Investigate the effect of different three-phase fluid compositions on flow characteristics within microchannels.
- Investigate the impact of dynamic flow conditions and fluctuations on the flow of three-phase fluid within microchannels.

1.3 SCOPE OF STUDY

The scope of our project encompasses a targeted exploration of microchannel design tailored for drug delivery applications. We will delve into the behavior of three-phase flow involving drug-carrying liquid, gas, and potentially solid particles within microchannels, considering factors such as channel geometry, dimensions, and surface properties. Our study will analyze drug dispersion, mixing characteristics, and heat transfer mechanisms to control drug release rates. Through the use of numerical simulations, particularly computational fluid dynamics (CFD), we aim to model and predict drug transport, optimizing microchannel parameters for targeted drug delivery efficiency. Validation against experimental data will be pursued to ensure the accuracy of our simulation model. We will explore the integration of the three-phase flow system with microfluidic devices for real-time monitoring and feedback control. Additionally, our analysis will extend to the efficiency of drug delivery systems, considering controlled release, dosage accuracy, and the minimization of side effects. Comparative studies against traditional two-phase flow systems will highlight advantages and limitations, while addressing regulatory and ethical considerations associated with novel microchannel designs for drug delivery.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION TO MICROFLUIDICS AND THREE-PHASE FLOWS

The study of microfluidics focuses on the control of small fluid volumes, usually nanoliters or fewer, inside networks of channels with tens to hundreds of micrometers in size. Microfluidics studies manipulating and controlling fluids at the micrometer scale, with applications in biomedical research, drug delivery, chemical synthesis, and environmental monitoring. Devices with small dimensions, large surface-to-volume ratio, and low Reynolds number simplify fluidic control methods. Further benefit of such small dimensions is the substantial reduction in sample and reagent volumes, which results in lower reagent costs and less waste. A platform with a tiny footprint can be produced by integrating several parts and procedures onto a single device (Tarn & Pamme, 2014)

The gas-liquid-liquid (G/L/L) three-phase flow in microchannels is crucial in various fields, including materials, chemistry, biology, and engineering. It was first applied to protein crystallization in 2005, and has been used in chemical reaction synthesis, core-shell structure, and porous structure. In materials, it is used to synthesize inorganic titanium or silicone hollow particles and porous hydrogel microspheres. The G/L/L system has also been widely applied to extraction, with the introduction of inert gas intensifying mass transfer and improving extraction efficiency. Two primary microchannel structures are used to prepare and synthesize three-phase flow: single dispersion units and combined dispersion units. Single units offer simplicity, easy manufacture, and practicality, while combined dispersion units are easier to regulate and control, allowing for material addition (Chen et al., 2019).

2.2 NUMERICAL SIMULATION TECHNIQUES

Using the VOF approach, the liquid-liquid two-phase flow in a T-junction was explored numerically and the results are compared with experiments. The commercial program Fluent (Fluent 12) was used to solve the appropriate equations for the CFD analysis after the geometry was created and meshed using the Gridgen program. Hexahedral elements in block-structured grids make up the produced mesh. For this study, fluids that are to be dispersed (water-glycerol solution) and continuous (silicone oil) at room temperature are taken into consideration. The impact of multiple factors, including surface tension, viscosity, width of the channel, and phase flow rate, on the droplet formation is examined and contrasted with existing experimental findings.

It is explained how droplets split apart in different capillary-number regimes. Based on the same geometrical and physical features, the experimental values and the numerical results of the length of the produced droplets as a function of the capillary number (changing the flow rate of the continuous phase) correspond well. Research reveals that when the disperse phase flow rate increases at a constant continuous phase flow rate, the droplet length increases significantly. On the other hand, the relative effects of the surface tension between phases and the continuous phase's viscosity on droplet length are moderate insight into droplet splitting in different capillary-number regimes (Shobeir Aliasghar Zadeh,2011).

Both the finite element and finite volume methods successfully approximate the analytic solution of two-cylinder domains in two dimensions. The main difference lies in the linear equation system. Direct solvers provide high accuracy for finite elements, while iso-parametric methods offer good convergence rates. However, direct solvers are not suitable for 3D problems due to memory consumption. Open-FOAM solves the steady state equation sequentially using a time-stepping method, decoupling heat transport from divergence-free constraints. For sin-equations, standard iterative solutions are employed. Poor temperature convergence and dubious stability for complicated issues are among the drawbacks of Open-FOAM (Westerkamp et al., 2014).

2.3 CROSS JUNCTION MICROCHANNELS IN DRUG DELIVERY SYSTEMS

Drug delivery systems are crucial for the treatment of disease, and nano- and microparticles offer advantages including stability and targeting. A possible method to create monodispersed particles with customizable physical properties is droplet microfluidics technology. This advancement in drug delivery may improve treatment outcomes while lowering adverse effects.

Al-wdan and colleagues addressed the challenges related to controlling particle size and shape by examining the use of microfluidics in particle production. Microfluidic systems can be utilized to accomplish controlled medication release, precise dosing, targeted distribution, and drug-loaded particles. They can produce materials including lipid-based, polymer-based, and inorganic particles. Biopolymers, such liposomes and chitosan, are popular because to their excellent biocompatibility and drug-loading capability. This makes it possible to administer several medications at once, which helps with combination therapy. Controlling the release of treatments at specific places is part of drug delivery. To release medications at a predefined rate and location, stimuli-responsive components can be added to drug-loaded particles.. This enhances selectivity and precision, improving treatment outcomes and reducing side effects. Microfluidic devices offer a platform for developing programmable drug delivery systems (Al wdan ,2023).

CHAPTER 3

METHODOLOGY

3.1 GEOMETRY

In microchannel geometry, the cross-flow junction is essential for managing three-phase flow dynamics. This intersection guides the convergence of different fluid phases in confined microscale spaces, impacting phase distribution, pressure drops, and overall system performance. Understanding fluid behavior at this point is crucial for optimizing microchannel devices.

In our project, we are using the following cross junction (T junction) microchannel geometry from a reference journal (Wang et al. 2013) as a model for our simulation. Selection of T-junction helps in designing of efficient phase separation and also this geometry can analyses how the pressure changes which helps in the optimization of design to minimize any such losses. By accurately modeling flow dynamics at a T-junction, engineers can make informed decisions about system design, operation, and maintenance, improving the efficiency and reliability of multiphase flow applications

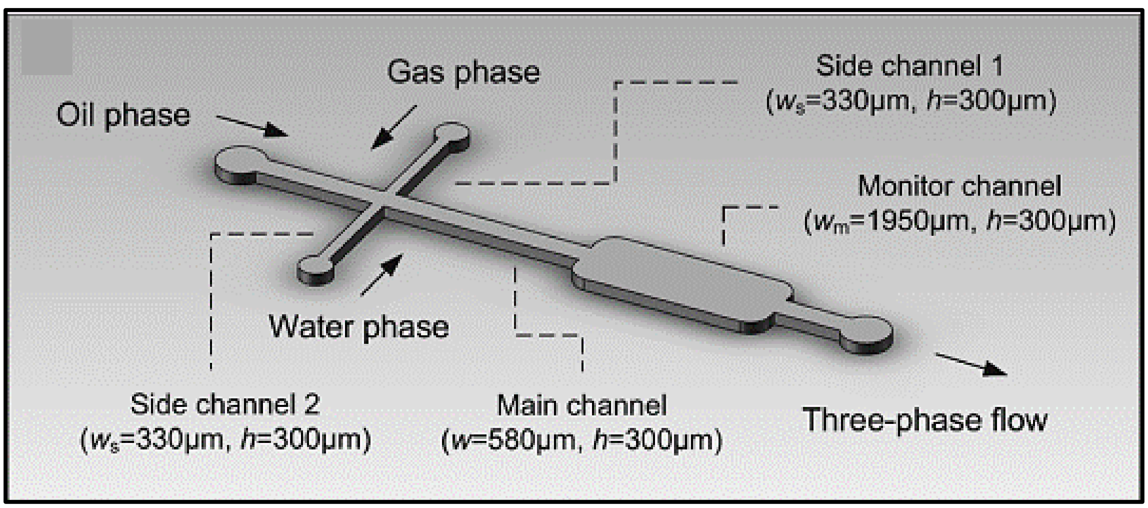


Fig 3.1: 3D structure of the microchannel

The feed pressure, is kept between 0.005 and 0.055 MPa (gauge pressure). Here, we are studying a three-phase flow: the continuous phase is oil, which is 60% paraffin mixed oil with 1wt% span 85 as surfactant; the third phase is air. The liquid dispersion phase is deionized water

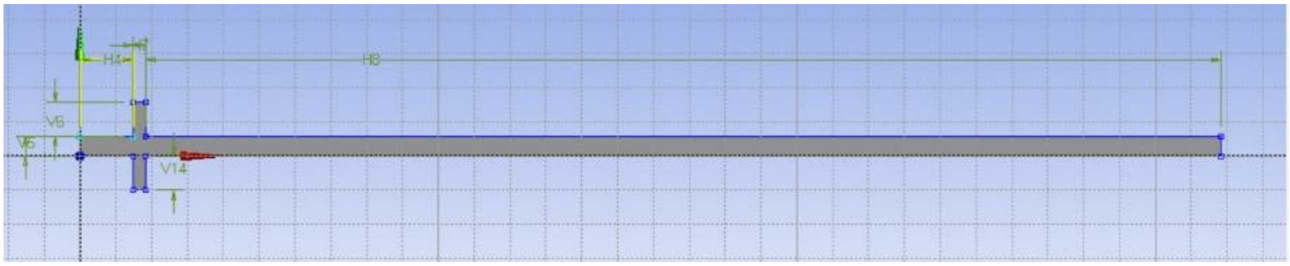


Fig 3.2: Geometry of Cross Junction Microchannel

As seen in the above diagram, phase 1 (oil) is provided through the main channel, while phase 2 (air) and phase 3 (water) are supplied by side channels 1 and 2, respectively, maintaining velocity at the microscale.

3.2 MESHING

Meshing is a critical step in the process of Computational Fluid Dynamics (CFD) analysis. It involves the discrete representation of the geometry of the problem. In this project, we have used ANSYS ICEM CFD for the meshing process.

3.2.1 Overview

Ansys ICEM CFD is a crucial pre-processing tool for Computational Fluid Dynamics mesh production. Numerous mesh types are supported, including prism, quad, triangle, hexahedral, and tetrahedral meshing. The choice of a specific mesh type in Ansys ICEM is dependent on multiple parameters, mostly determined by the simulation specifications and geometric complexities of the model.

Hexahedral meshing is the method of choice when precise geometry representation and effective computational performance are essential. Conversely, tetrahedral meshing works well in complex geometries where it might be difficult to create hexahedral pieces. For boundary layer resolution, prism/hexa meshing achieves a balance that is frequently beneficial. Surface meshing, either quad dominating or triangular, can be selected depending on the elements required and the physics of the simulation. For irregular geometry, unstructured meshing allows flexibility, but structured meshing makes certain simulations simpler. The particular requirements of the simulation scenario serve as a reference for the trade-off that must be made during the selection process between accuracy, computing efficiency, and mesh production simplicity. The quad meshing method is the one used in our project.

3.2.2 Quad Meshing

- Geometry Blocking: To begin, separate the sketch into several blocks that will be independently meshed. To create the block, first initialize the blocks by selecting '2D Planar' as the type of the block.
- Associating Entities to the Geometry: The vertices, edges, and faces of the blocks are associated with the corresponding entities of the geometry.
- Applying Mesh Parameters: In the blocking section, we need to select 'Pre-Mesh Parameters'. Here we are using 'Edge Parameters' to select the edges of the sketch to divide it into nodes using the 'Bi-Geometric' mesh law where we specify values for spacing 1 and spacing 2 which define size of the first and last element along the edge respectively. Next, we have to specify Ratio 1 (Ratio of size of second cell to that of first cell) and Ratio 2 (ratio of size of last cell to that of the second last cell) which help to define the growth rate of the mesh along the edge. It is important to specify the 'Max Size' to control the maximum allowable size of an element in the mesh
- Generating the Initial Mesh: The initial mesh is generated based on the specified parameters. Initially, the mesh may not be perfect. But it can help us to refine the mesh since we have to create a denser and more complex mesh at the walls of the channel to account for the boundary layer developing near the walls and allow the governing equations to be solved with better accuracy, particularly in these regions.
- Refining the Mesh: The mesh is refined to improve its quality. The mesh info gives us valuable information such as element types, element parts, and also the total number of elements and nodes. By varying any of the specifies pre-mesh parameters, we can vary the number of elements of the mesh and thereby improve mesh quality.

- Verifying and Saving the Mesh: The final step involves verifying the quality of the mesh and saving it.

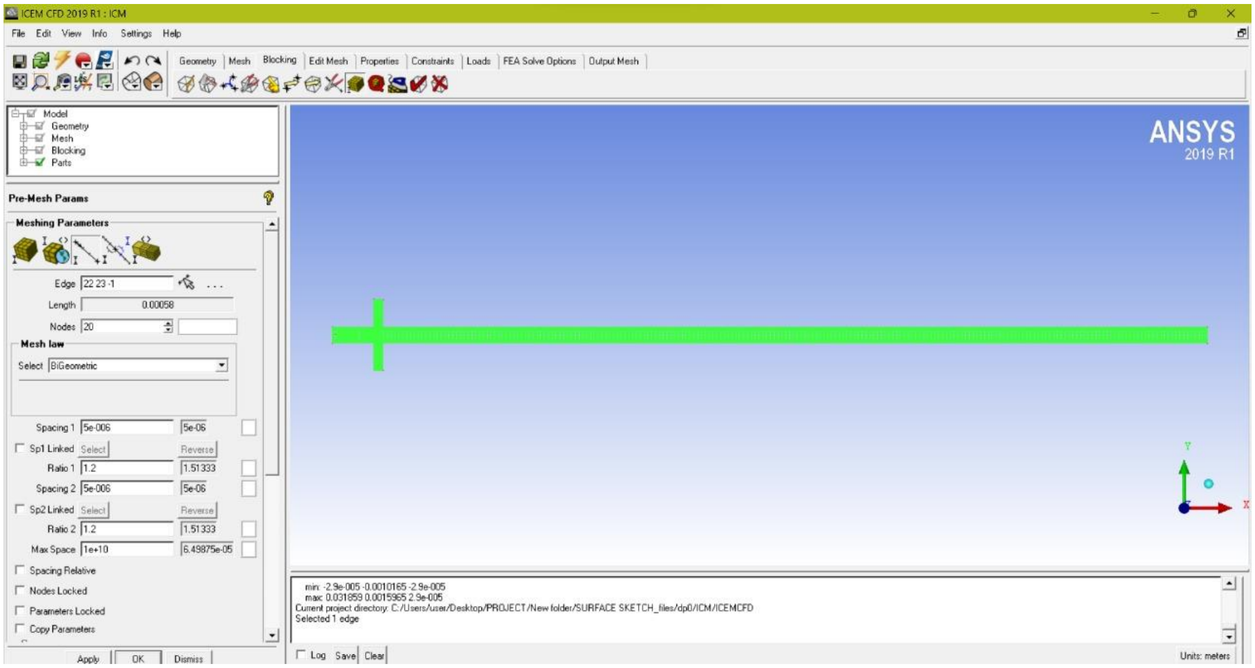


Fig 3.3: Geometry of cross junction microchannel after meshing

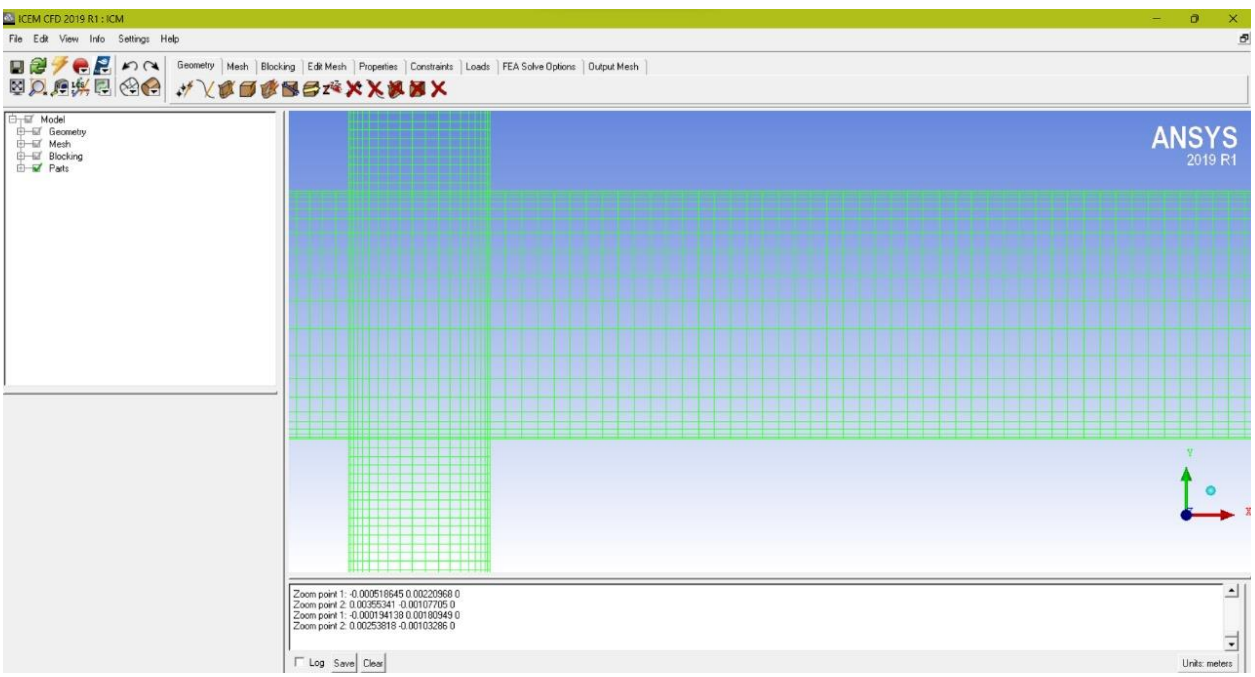


Fig 3.4: Magnified image of meshed microchannel

3.3 PHYSICAL MODELS AND ASSUMPTIONS

3.3.1 Governing Equations of Conservation

Interface tracking is an essential phenomenon to gain insights into the complex behavior of flow and for the optimization of its performance. Tracking of interface can be done in several ways, which include, moving grid, tracking marker particles and volume of fluid method. In order to track the interface between two immiscible phases, a Eulerian method called Volume of Fluid method is employed. The tracking is achieved by solving the volume fractions of each phase in each computational cell.

$$\frac{\partial \alpha_i}{\partial t} + \nabla.\ \alpha_i u_i = \sum_{j \neq i} \Gamma_{ij}(\alpha_j - \alpha_i)$$

For the provision of a solid foundation to represent physical phenomena, to enable quantitative predictions and for the optimization of design, it is essential to solve certain governing equations. The governing equations employed in our project include continuity equation, navier-stokes equation and volume fraction equation.

(i) Continuity equation:

$$\frac{\partial(\alpha_i \rho_i)}{\partial t} + \nabla.(\alpha_i \rho_i \bar{u_i}) = S_i$$

where ρ_i : density of phase i

α_i : volume fraction phase i

u_i : velocity of phase i

S_i : source or sink terms for phase i, such as phase change or chemical reactions

(ii) Navier-Stokes' Equation:

$$\rho_i \left(\frac{\partial v_i}{\partial x} + v_i.\ \nabla v_i\right) = -\nabla p_i + \nabla \tau_i + f_i$$

where, ρ_i : density of phase i

v_i : velocity of phase i

p_i : pressure of phase i

τ_i : stress tensor of phase i

f_i : external forces acting on phase i

(iii) Volume Fraction Equation:

$$\frac{\partial \alpha_i}{\partial t} + \nabla.\ \alpha_i u_i = \sum_{j \neq i} \Gamma_{ij}(\alpha_j - \alpha_i)$$

where, Γ_{ij} : volume fraction flux coefficient between phases i and j

3.3.2 Assumptions

- In order to avoid divergence in our simulation and for better accuracy, we have utilized the Implicit method which provides a constant Courant number with variable times step.
- The volume fraction cutoff is given as 1×10^{-6}

3.4 SETUP

Setting up a simulation in a microchannel involving three phases is a critical aspect of microfluidics research. This simulation aims to model the intricate interactions among different fluid phases within confined microscale environments.

Our simulation is conducted using Finite Volume based solver ANSYS FLUENT. A pressure-based solver type is used in a 2D spaced planar model with time being in transient phase.

To make calculations easier, the phase flowrates are kept at the microscale. Variations in velocity between 10 and 600 microliter/min are used to maintain the oil phase flowrate in the main channel, and between 3 and 500 microliter/min are used to maintain the water phase flowrate in the side channel. Gauge pressure is varied between 0.005 and 0.05 MPa to maintain the air phase on the other side channel.

3.4.1 Physical Properties of the working system (25 °C)

The simulation is operated at a room temperature of 25°C and the following table depicts the numerical values used for the physical properties of the working system:

Table 3.1: Physical Properties of working system

Phases	**Fluid**	**Viscosity** μ **[mpa.s]**	**Densities** $\dot{\rho}$ **[kg/m^3]**	**Interfacial Tension** Γ **[mn/m]**
Oil	1 wt % span 85 Paraffin solution	44.9	854	-
Gas	Air	0.018	1.165p + 0.116	22.3
Water	Deionized water	0.9	997	7.8

3.4.2 Boundary Conditions

The boundary conditions used in the simulation are listed in table 3.2. Here, the inlet parameters allotted for each phase are converted from their flowrates to respective velocities. The wall is at no-slip condition.

The critical insights on the impact of variations in contact angle in the microfluidic study of multiphase flow within microchannels have been discussed. For instance, the slug flow is seen to predominate at a range of contact angles between 0^0 to 30^0 . On increase in the contact angles further to 90 the flow transformed to annular flow in which liquid form thin film along channels of wall surrounded by gas phase. If we further rise the angle beyond 90 we could see a prominent stratified flow in which various layers of miscible liquids become visible distinctly. Contact angle variations not only effect phase distribution but also impact the pressure drop characteristics and heat exchanger rates. Since when we started digging deep to this parameter impacts we could study the droplet breakup and coalescence deeply affected by this .

.**Table 3.2:** Inlet parameters of various flow modes

Sl No.	Flow Mode	Velocity of air (Ua) (m/s)	Qw (m3/s)	Qo (m3/s)	Velocity of water (Uw) (m/s)	Velocity of oil (Uo) (m/s)
1	Bubble cutting	0.046	1.17E-10	8.33E-09	1.18E-02	4.79E-03
2	Jetting and cutting	0.046	1.67E-09	2.50E-09	1.68E-02	1.44E-02
3	Laminar	0.035	5.00E-09	5.00E-09	5.05E-02	2.87E-02
4	Multi-bubble cutting	0.046	3.33E-10	1.67E-09	3.37E-03	9.58E-03
5.	Laminar and interface shearing	0.035	1.67E-09	1.67E-09	1.68E-02	9.58E-03

Table 3.3: Contact angle and surface tension coefficient between phases

Phases	Contact angle (º)	Surface Tension Coefficient (N/m)
oil (1)-water (3)	105	0.03
oil (1)-air (2)	95	0.02
air (2)-water (3)	90	0.0728

3.5 SOLUTION

A CFD analysis solution step involves solving the equations governing the fluid flow for the specified boundary conditions and mesh. The pressure-velocity coupling scheme utilized is SIMPLE. This algorithm is renowned for its ability to iteratively solve for pressure and velocity fields, resulting in highly accurate and stable outcomes.

3.5.1 Solver Initialization

Before starting the solution process, the solver needs to be initialized. This involves setting initial guesses for the flow variables. In ANSYS Fluent, the standard initialization method is selected which initializes the flow field using a uniform state. Then, we have to specify the initialization zone, which is the whole domain.

3.5.2 Running the Solver

Once the solver is initialized, the next step is to run the solver. This involves starting the iterative solution process. The solver will continue to iterate until the change in the solution between iterations is below a specified threshold. Before engaging the solver, we mainly have to specify time stepping method, time step size, max. iteration/time step and reporting interval.

- Time Stepping Method: It allows the us to choose how the time step is determined for transient flow calculations. The 'Fixed' mode of time stepping method is selected so that we can specify constant time step size.
- Time step size: A time step is a small duration of time used to advance the simulation forward, during which the state of the flow throughout the entire area being studied is calculated. A smaller time step size can lead to a more accurate solution, but it will also require more iterations and thus more computational time. Conversely, a larger time step size can speed up the calculation, but it may also reduce the accuracy of the solution or cause the solution to diverge.
- Max iteration/time step: It is the maximum number of iterations that the solver will perform for each time step. If the solution for a particular time step converges before this number is reached, the solver will move on to the next time step.
- Reporting Interval: It denotes the number of time steps between each report of the solution progress. More regular updates on the solution's progress will be provided by a shorter reporting interval, but the reporting overhead may cause the computation to lag.

3.5.3 Monitoring Convergence

Keeping an eye on the solution's convergence is crucial while the solver is operating. Residuals, or the difference between the left and right sides of the governing equations, are what we mostly employ and they should get smaller with each repetition. A potential issue with the solution procedure could be indicated if the residuals are not declining.

3.6 RESULT

The primary objective of our simulation was to shed light on the phase distribution patterns and the associated flow dynamics of the system involving three phase flow through microchannels. Through our study, we were able to detail the intricate working of such a system where three phase flow through a microchannel occurs.

3.6.1 Contour Pattern of Phase ID:

Unique phase identification (ID) contour patterns where able to be observed inside the microchannel through detailed analysis of the simulation. The colour coded phase ID contours served as an essential tool in the depiction of the interactions and influences between the liquid and gas phase and their corresponding geographical distributions in the microscale environment. The location of areas where phase mixing, separation or possible association or breakage occurs where able to be sorted outed using the contours.

a) Number of elements = 40,000

The general properties of flow were analyzed using early simulations with 40,000 meshes.

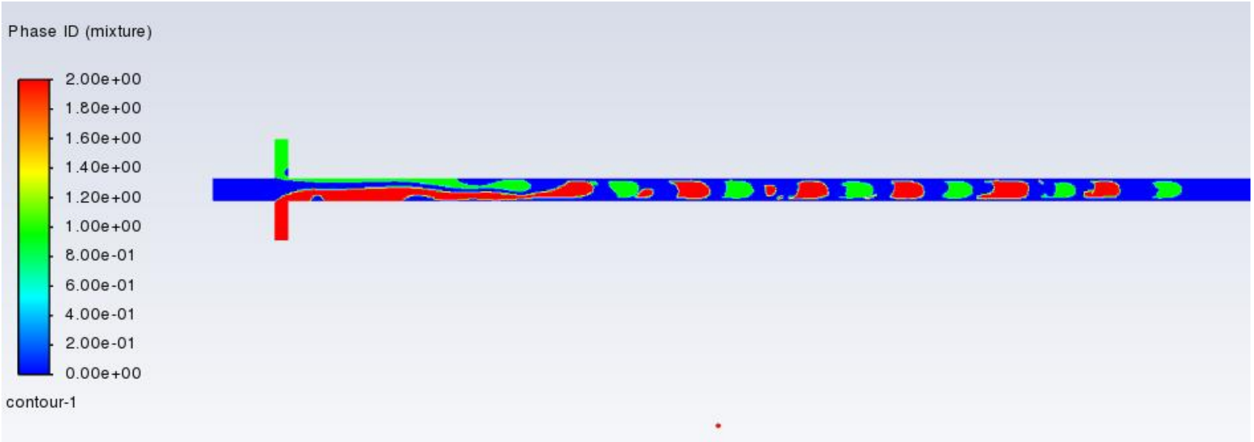


Fig 3.5: Number of elements =40000

b) 80,000 Meshes:

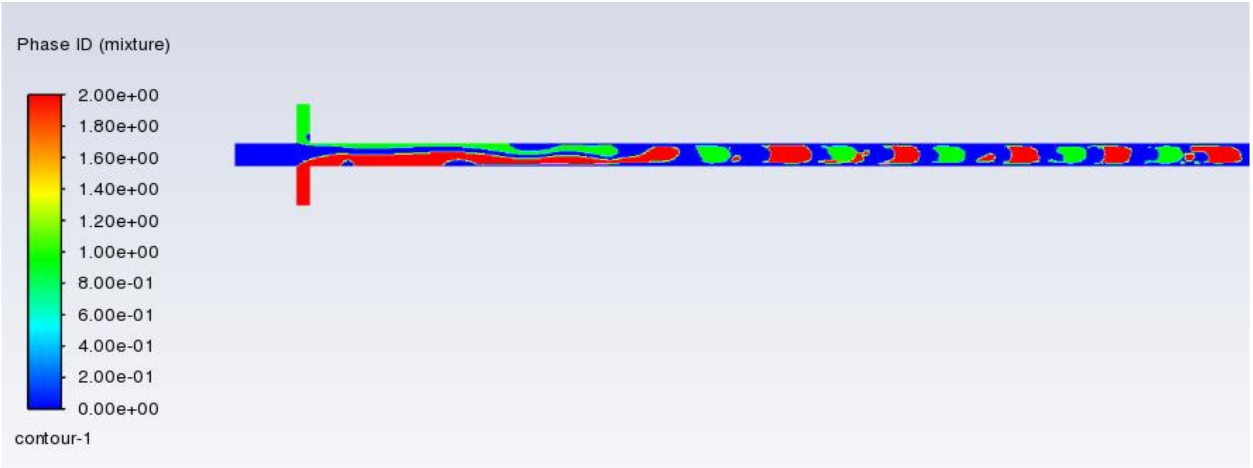


Fig 3.6: Number of elements = 80000

3.6.2 Mesh Independency Analysis:

The mesh sensitivity (also known as a mesh independence or mesh convergence study) study is an important tool to verify the accuracy of results of the CFD (Computational Fluid Dynamics) simulations. After splitting the computational domain into discrete mesh cells, the governing flow equations are applied and the results are calculated using the mesh as the base. This will lead to the origination of spatial discretization errors and by running mesh sensitivity study, the user can have a better understanding of the spatial discretization errors and how much the mesh is affecting the solution. Practically, when we need the results to be mesh independent, they should change by only a fine amount with the additional refinement. The mesh sensitivity study consist of running the same simulation using meshes of different resolutions to ensure that solution is independent of the resolution of the mesh and it helps to analyze how much the converged solution changes with each mesh resolutions.

Using Ansys ICEM-CFD we perform the meshing process, starting by creating a coarse mesh to divide the microchannel into small, discrete cells. Then after meshing, we solve the flow equations using Ansys Fluent. We have to monitor the scaled residuals and contour of phases. Next, we refine the mesh by decreasing the element size, which increases the number of cells from 40,000 cells to 80,000 cells and the simulation was repeated.

1. Residual plot

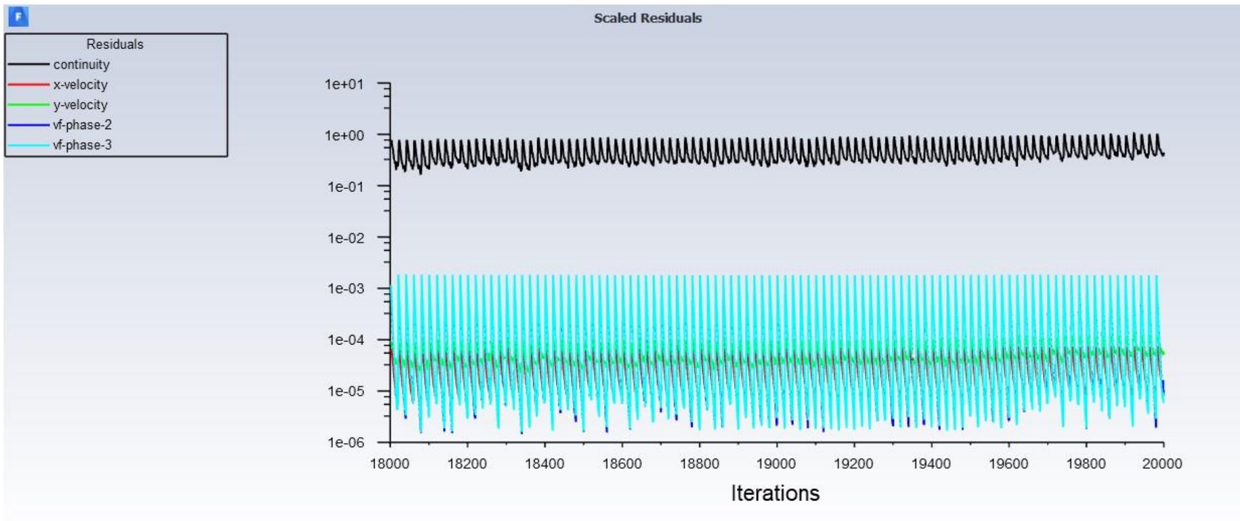


Fig 3.7: Number of elements = 40000 after a time step of 1000

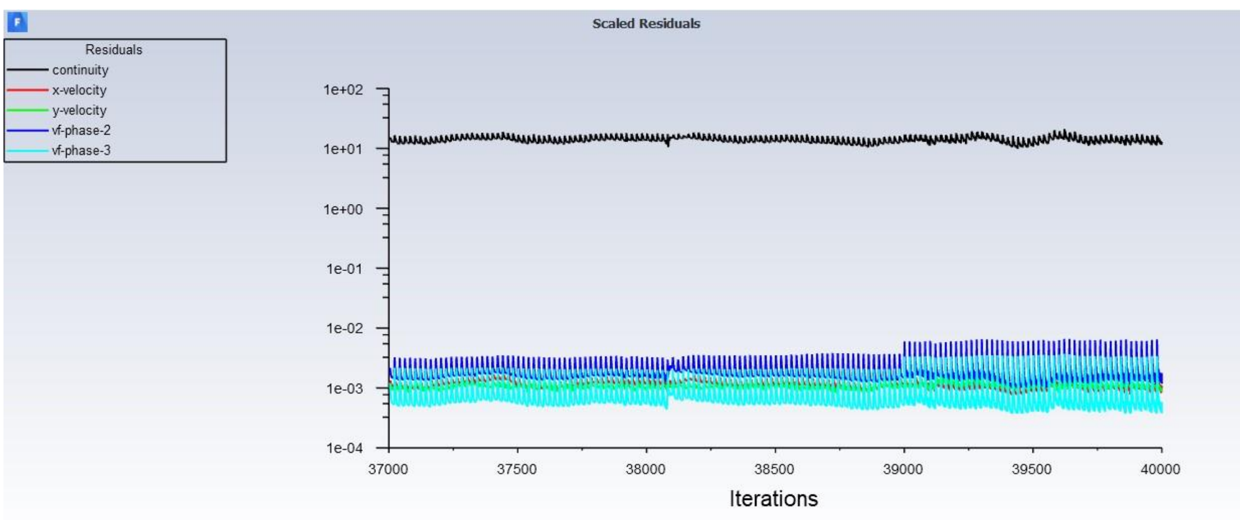


Fig 3.8: Number of elements = 40000 after a time step of 2000

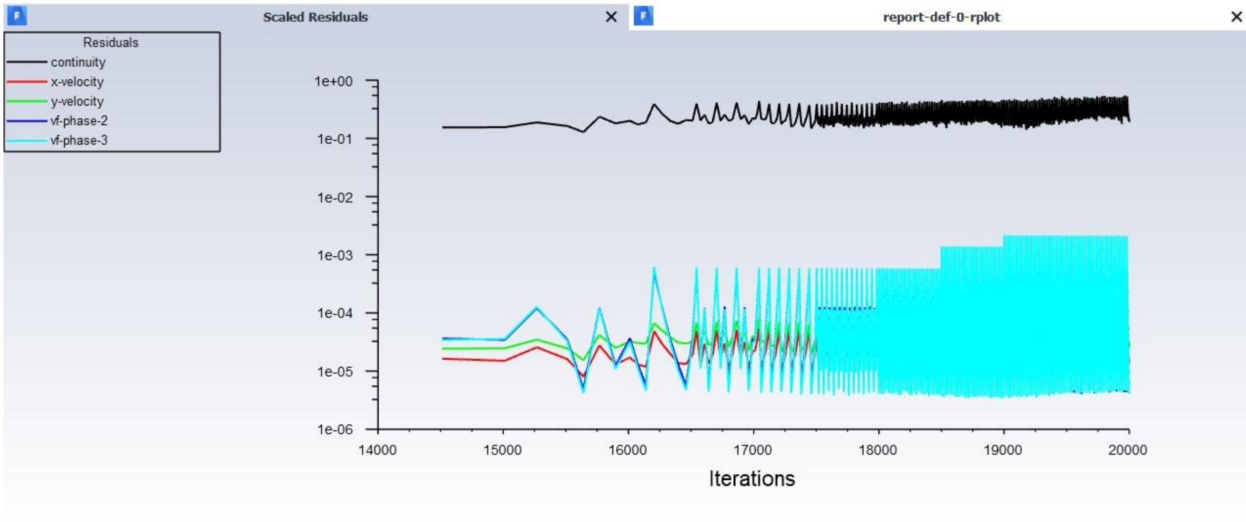


Fig 3.9: Number of elements = 80000 after a time step of 1000

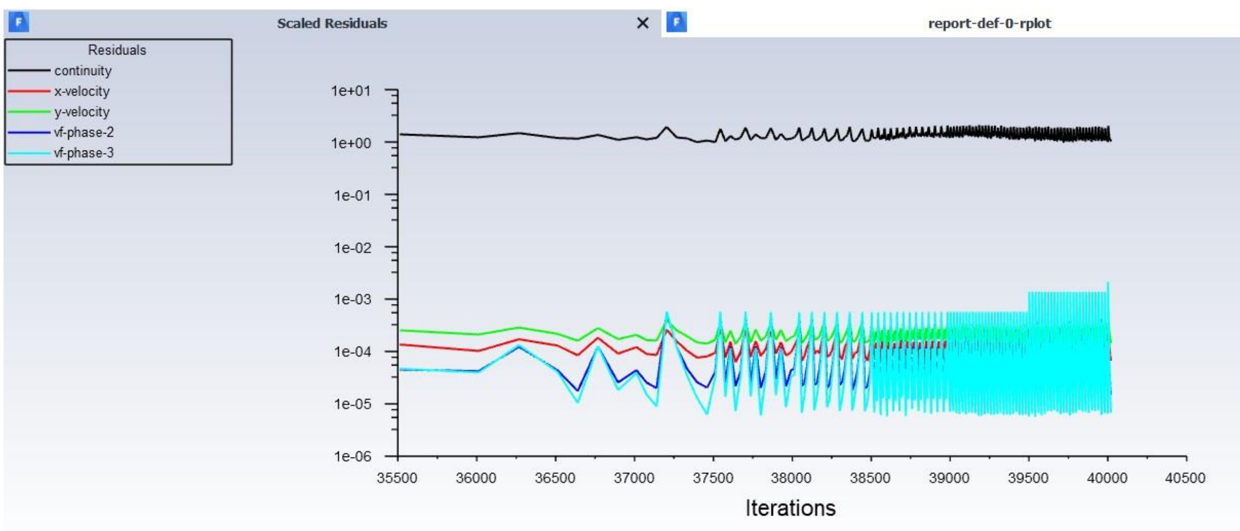


Fig 3.10: Number of elements = 80000 after a time step of 2000

The results obtained from the meshes were compared and the results were found to be without much difference, thereby indicating the solution to be mesh independent.

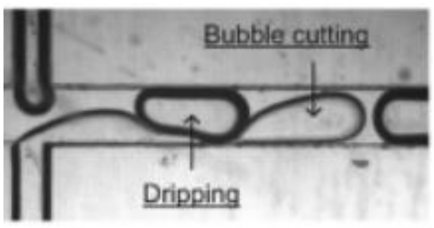
3.6.3 Comparison of Experimental Results with Simulated Results

1. Bubble Cutting

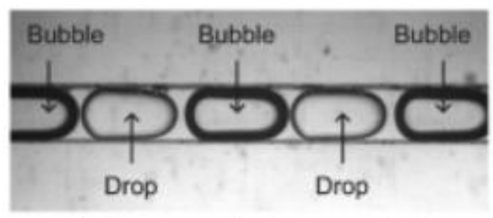
In multiphase microchannel flow, bubble cutting occurs as gas bubbles interact with channel walls. Key parameters affecting this include flow rates (e.g., gas and liquid velocities), fluid properties (e.g., viscosity, surface tension), and channel dimensions (e.g., width, height). The numerical values influence bubble deformation, fragmentation, and mixing within the microchannel, ultimately impacting overall flow dynamics.

(a) Experimental Result:

Cross-flow Junction

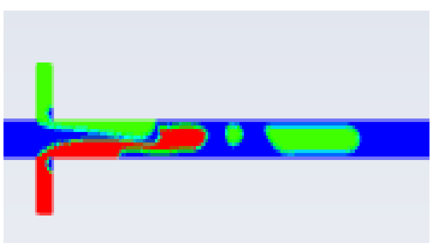


Main channel



(b) Simulation Result:

Cross-flow Junction



Main Channel

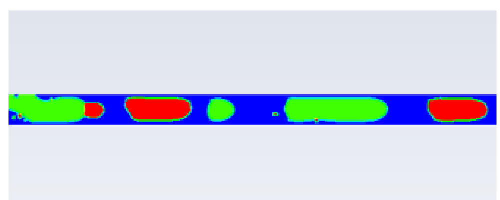


Fig 3.11: Validating simulated results (b) with experimental (a) in the bubble cutting flow regime

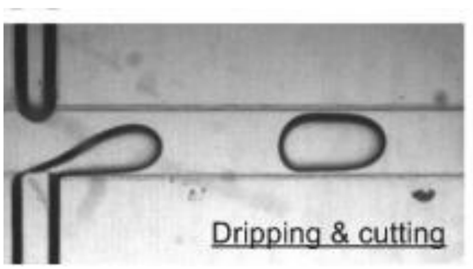
2. Jetting and Cutting

Jetting is a condition wherein the jets of high velocity stream occur within other phases. In the three-phase flow scenario, from the input parameters we can see that the velocity of oil is greater than water, due to which it penetrates the water or air phase at the junction.

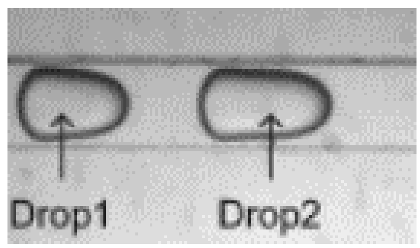
Cutting is a phenomenon in which one phase displaces the other by effectively cutting through it. In our simulation, the droplets are cut into many leading to the flow pattern of jetting and cutting. Through our simulation, we can clearly understand these cutting events by tracking the interface between phases and depicting how the oil phase penetrates and displaces other phases within the junction

(c) Experimental Result:

Cross-flow Junction

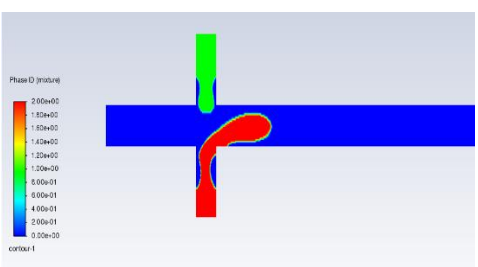


Main channel



(d) Simulation Result:

Cross-flow Junction



Main Channel

Fig 3.12: Validating simulated results (b) with experimental (a) in the jetting and cutting flow regime

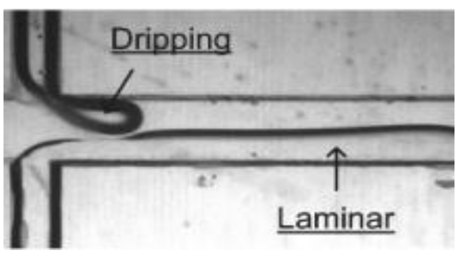
3. Laminar flow

Dripping is a condition when one phase form droplets that detach from the main channel. It mainly occurs when one parameter is greater than the other. Here, the gas phase is having greater value than other phases due to which the growing bubble does not block the main channel.

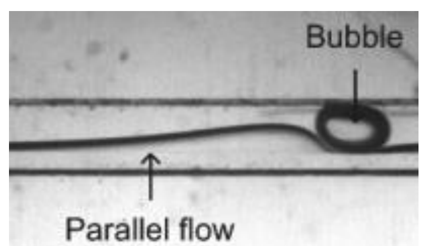
Laminar flow indicates the smooth and ordered pattern of flow that moves parallel and has no or minimal disruption between those layers. It is mainly found to occur when layers of each phase are distinct and the flow is found to be stable. In our context, the input value of flowrate for water and oil phases are found to be same which had resulted in a stable flow indicating the laminar flow regime.

(a) Experimental Result:

Cross-flow Junction

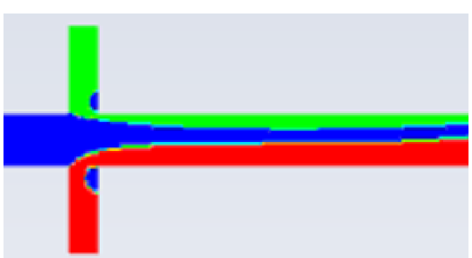


Main channel



(b) Simulation Result:

Cross-flow Junction



Main Channel

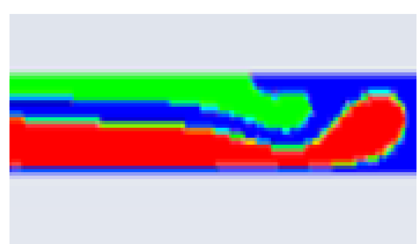


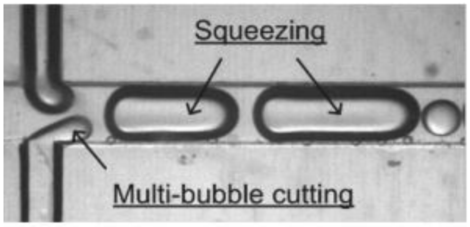
Fig 3.13: Validating simulated results (b) with experimental (a) in the laminar flow regime

4. Multi-bubble Cutting

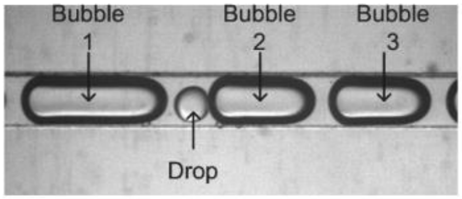
Multi-bubble cutting refers to a phenomenon where a gas bubble is split into multiple smaller bubbles due to interactions with the channel walls or other phases in multiphase flow through microchannels. This process typically involves the deformation and fragmentation of the original bubble into multiple separate entities.

(a) Experimental Result:

Cross-flow Junction

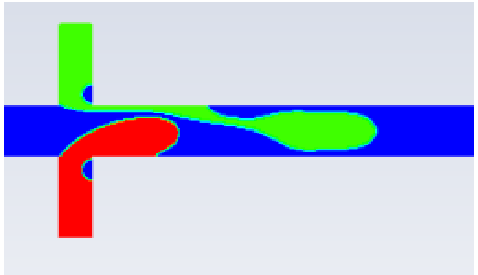


Main channel



(b) Simulation Result:

Cross-flow Junction



Main Channel

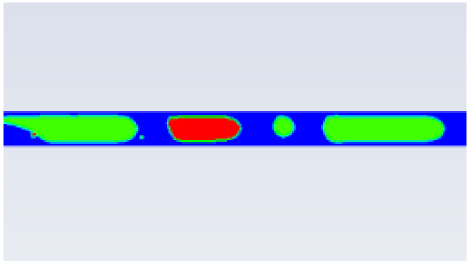


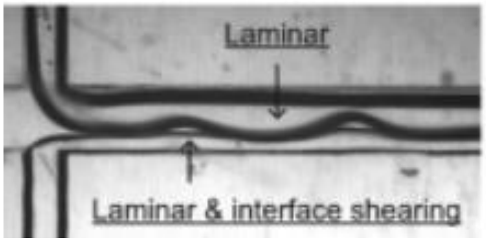
Fig 3.14: Validating simulated results (b) with experimental (a) in the multi-bubble cutting flow regime

5. Laminar and Interface Shearing

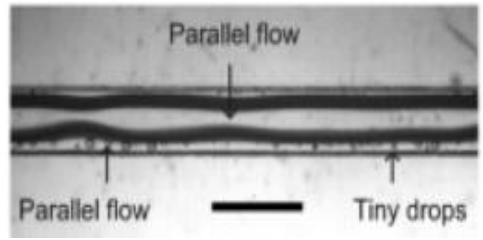
Interphase shearing refers to the relative motion or sliding between two adjacent phases in multiphase flow systems. In the context of microchannel flows, it specifically describes the movement between different phases, such as gas-liquid or liquid-liquid interfaces, as they flow through the channel. This shearing action occurs due to the velocity difference between the phases, which results in stress and deformation at the interface.

(c) Experimental Result:

Cross-flow Junction

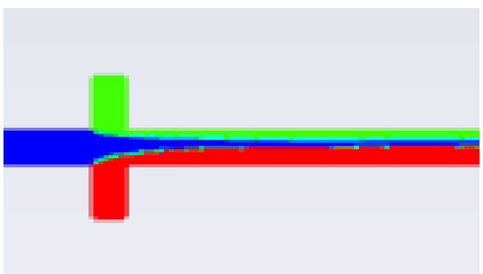


Main channel



(d) Simulation Result:

Cross-flow Junction



Main Channel

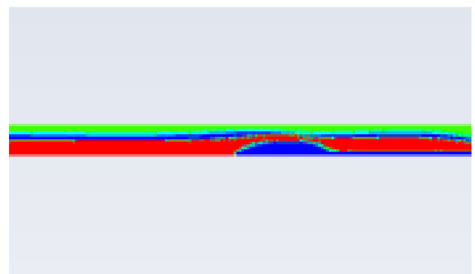


Fig 3.15: Validating simulated results (b) with experimental (a) in the laminar and interface shearing flow regime

CHAPTER 4

CONCLUSIONS

In conclusion, our study was able to demonstrate the important role played by the interactions between drug carrying liquid, gas and possible solid particles in the overall dynamics of flow. This in turn was made possible through the numerical simulation of three phase flow in microchannels. The influence of geometry was found to be significant on the flow patterns, which leads to the finding that the key to better performance in drug delivery systems is to optimize these structures. Validation of our numerical model was done by comparing with experimental data, which validated its accuracy and reliability in analyzing three phase flow through microchannels. The area of research of examining various operating conditions and fluid characteristics to improve design and optimization of microscale drug delivery system is astonishing and full of possibilities. There is a huge space for development and deeper scope for understanding and acquiring knowledge.

REFERENCES

1. Al-wdan, O. A., Sharallah, O. A., Abdelwahab, N. A., Mohammed, A. O., Elmowafy, E., & Soliman, M. E. (2023). Insights into microfabrication and implementation of microfluidics in pharmaceutical drug delivery and analysis. *Open Nano*, 12, 100156
2. Cao, Zhou, Yu, & Liu. (2019, February). Droplet-based mixing characteristics in bumpy serpentine microchannel. *Chemical Engineering and Processing - Process Intensification*, *159*. https://doi.org/10.1016/j.cep.2020.108246
3. Chen, Zhou, Wang, & Xu. (2019). Regulation and scaling law of gas-liquid-liquid three-phase flow in a dual-coaxial microchannel. *Chemical Engineering Science*, *209*. https://doi.org/10.1016/j.ces.2019.115172
4. D. Hernández-Cid, V.H. Pérez-González, R.C. Gallo-Villanueva, J. González-Valdez, and M.A. Mata-Gómez. 2022. “Modelling Droplet Formation in Microfluidic Flow-Focusing Devices Using the Two-Phases Level Set Method.” *Materials today proceedings* 48. https://doi.org/10.1016/j.matpr.2020.09.417.
5. D. Tice, Song, D. Lyon, & F. Ismagilov. (2003, August 12). Formation of Droplets and Mixing in Multiphase Microfluidics at Low Values of the Reynolds and the Capillary Numbers. *Langmuir*, *19*(22), 9127–9133. https://doi.org/10.1021/la030090w
6. Erbil, H.Y. 2021. “Dependency of Contact Angles on Three-Phase Contact Line: A Review.” *Colloids and Interfaces* 5 (1) (February 1):8. doi:10.3390/colloids5010008. http://dx.doi.org/10.3390/colloids5010008.
7. Fan, Ma, Wang, Luo, & Komarneni . (2021, October). Gas-liquid-liquid flow patterns and extraction in a rotating microchannel extractor. *The Canadian Journal of Chemical Engineering*, *99*(S1). https://onlinelibrary.wiley.com/doi/10.1002/cjce.23975
8. Gupta, R., D.F. Fletcher, and B.S. Haynes. 2009. “On the CFD Modelling of Taylor Flow in Microchannels.” *Chemical Engineering Science* 64 (12) (June):2941–50. doi:10.1016/j.ces.2009.03.018.

9. Jiang, & Zhou. (2020, November 4). Droplet behaviors on inclined surfaces with dynamic contact angle. *International Journal of Hydrogen Energy*, *455*(54), 29848–29860. https://doi.org/10.1016/j.ijhydene.2019.07.173
10. KÜÇÜKAKÇA MERAL, Z., and N. PARLAK. 2021. "EXPERIMENTAL RESEARCH AND CFD SIMULATION OF CROSS FLOW MICROCHANNEL HEAT EXCHANGER." *Journal of Thermal Engineering* 7 (2) (February 1):270–83. doi:10.18186/thermal.872366. http://dx.doi.org/10.18186/thermal.872366.
11. Ladosz, A., Rigger, E., & Rudolf Von Rohr, P. (2016). Pressure drops of three-phase liquid–liquid–gas slug flow in round microchannels. *Microfluidics and Nanofluidics*, *20*(3), 49. https://doi.org/10.1007/s10404-016-1712-7
12. Lei, L., Zhao, Y., Chen, W., Li, H., Wang, X., & Zhang, J. (2021). Experimental Studies of Droplet Formation Process and Length for Liquid–Liquid Two-Phase Flows in a Microchannel. *Energies*, *14*(5), 1341. https://doi.org/10.3390/en14051341
13. Li, W.-L., H.-W. Liang, J.-H. Wang, L. Shao, G.-W. Chu, and Y. Xiang. 2022. "CFD Modeling on the Chemical Absorption of CO2 in a Microporous Tube-in-Tube Microchannel Reactor." *Fuel* 327 (November):125064. doi:10.1016/j.fuel.2022.125064. http://dx.doi.org/10.1016/j.fuel.2022.125064.
14. Liu, & Zhang. (2009, August 1). Droplet formation in a T-shaped microfluidic junction. *Journal of Applied Physics*, *106*(3). https://doi.org/10.1063/1.3187831
15. Liu, D., X. Ling, and H. Peng. 2016. "Comparative Analysis of Gas–Liquid Flow in T-Junction Microchannels with Different Inlet Orientations." *Advances in Mechanical Engineering* 8 (3) (March 1):168781401663732. doi:10.1177/1687814016637329. http://dx.doi.org/10.1177/1687814016637329.
16. Madadelahi, M., & Shamloo, A. (2017). Droplet-based flows in serpentine microchannels: Chemical reactions and secondary flows. *International Journal of Multiphase Flow*, *97*, 186–196. https://doi.org/10.1016/j.ijmultiphaseflow.2017.08.010
17. Magnini, M., and J.R. Thome. 2016. "A CFD Study of the Parameters Influencing Heat Transfer in Microchannel Slug Flow Boiling." *International Journal of Thermal Sciences* 110 (December):119–36. doi:10.1016/j.ijthermalsci.2016.06.032. http://dx.doi.org/10.1016/j.ijthermalsci.2016.06.032.

18. Makarem, M.A., M. Farsi, and M.R. Rahimpour. 2021. “CFD Simulation of CO2 Removal from Hydrogen Rich Stream in a Microchannel.” *International Journal of Hydrogen Energy* 46 (37) (May):19749–57. doi:10.1016/j.ijhydene.2020.07.221. http://dx.doi.org/10.1016/j.ijhydene.2020.07.221.
19. Tarn, M. D., & Pamme, N. (2014). Microfluidics. In *Reference Module in Chemistry, Molecular Sciences and Chemical Engineering* (p. B9780124095472053518). Elsevier. https://doi.org/10.1016/B978-0-12-409547-2.05351-8
20. Thakur, N., & Murthy, H. (2022). Simulation study of droplet formation in inkjet printing using ANSYS FLUENT. *Journal of Physics: Conference Series*, *2161*(1), 012026. https://doi.org/10.1088/1742-6596/2161/1/012026
21. Wang, K., Lu, Y., Qin, K., Luo, G., & Wang, T. (2013). Generating Gas-Liquid-Liquid Three-Phase Microflows in a Cross-Junction Microchannel Device. *Chemical Engineering & Technology*, *36*(6), 1047–1060. https://doi.org/10.1002/ceat.201200561
22. Wang, K., Qin, K., Lu, Y., Luo, G., & Wang, T. (2015). Gas/liquid/liquid three-phase flow patterns and bubble/droplet size laws in a double T-junction microchannel. *AIChE Journal*, *61*(5), 1722–1734. https://doi.org/10.1002/aic.14758
23. Westerkamp, A., Bünger, J., & Torrilhon, M. (2014). *Simulation of Gas Micro Flows based on Finite Element and Finite Volume Method*.
24. Yang, S., G. Kong, Z. Cao, and Z. Wu. 2023. “Mass Transfer and Modeling of Deformed Bubbles in Square Microchannel.” *Chemical Engineering Journal Advances* 16 (November):100518. doi:10.1016/j.ceja.2023.100518.
25. Yin, & Kuhn. (2022, November 2). Numerical simulation of droplet formation in a microfluidic T-junction using a dynamic contact angle model. *Chemical Engineering Science*, *261*. https://doi.org/10.1016/j.ces.2022.117874
26. Yuan, Y., & Lee, T. R. (2013). Contact Angle and Wetting Properties. In G. Bracco & B. Holst (Eds.), *Surface Science Techniques* (Vol. 51, pp. 3–34). Springer Berlin Heidelberg. https://doi.org/10.1007/978-3-642-34243-1_1
27. Yue, J., Rebrov, E. V., & Schouten, J. C. (2014). Gas–liquid–liquid three-phase flow pattern and pressure drop in a microfluidic chip: Similarities with gas–liquid/liquid–liquid flows. *Lab on a Chip*, *14*(9), 1632. https://doi.org/10.1039/c3lc51307f
28. Zadeh, SA, & Radespiel, R. "Numerical Study on Droplet Formation in a Microchannel T-

Junction Using the VOF Method." *Proceedings of the ASME 2010 8th International Conference on Nanochannels, Microchannels, and Minichannels collocated with 3rd Joint US-European Fluids Engineering Summer Meeting*. *ASME 2010 8th International Conference on Nanochannels, Microchannels, and Minichannels: Parts A and B*. Montreal, Quebec, Canada. August 1–5, 2010. pp. 1601-1610. ASME. https://doi.org/10.1115/FEDSM-ICNMM2010-30191

29. Zhuo Chen, Yi-Wei Zhou, and Yun-Dong Wang. 2019. “Regulation and Scaling Law of Gas-Liquid-Liquid Three-Phase Flow in a Dual-Coaxial Microchannel.” *Chemical Engineering Science* 209 (December 14). doi: https://doi.org/10.1016/j.ces.2019.115172.