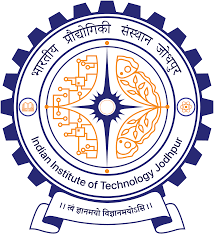
**DESIGN CREDIT PROJECT REPORT**

**DESIGN AND SIMULATION OF A MICROFLUIDIC DROPLET GENERATOR WITH FLOW FOCUSING GEOMETRY IN ANSYS**



**SUBMITTED TO:**

**PROF.TARA CHAND KUMAWAT**

**SUBMITTED BY:**

**AYUSHI SINGH (B22CH005)**

**ABHISHEK KUMAR (B22CH001)**

**APURVA JALWANIYA (B22CH003)**

**ABSTRACT**

This project presents the simulation of a microfluidic droplet generator using flow-focusing geometry in ANSYS Fluent. Two immiscible fluids—water and coconut oil—were modeled using the Level Set method to analyze droplet formation dynamics. Results showed that droplet size decreased with increasing continuous phase velocity, confirming the inverse relationship. The simulation successfully replicates experimental behavior, making it valuable for designing Lab-on-a-Chip systems**.**

**AIM**

To design and simulate a microfluidic droplet generator based on flow-focusing geometry using ANSYS Fluent, with the goal of understanding the droplet formation mechanism, optimizing geometric parameters, and analyzing the effects of flow rate variations on droplet characteristics for lab-on-a-chip applications.

**OBJECTIVES**

* To model a 2D microchannel system mimicking a flow-focusing droplet generator.
* To define and assign material properties for two immiscible fluids (e.g., water and coconut oil) in the simulation.
* To implement a multiphase flow simulation using the **Level Set** or **Volume of Fluid (VOF)** method in ANSYS Fluent.
* To simulate droplet formation under varying flow rate ratios and analyze the effect on droplet size and detachment behavior.
* To validate simulation results against existing experimental observations and literature (e.g., Hettiarachchi et al., 2021).
* To visualize the droplet generation process using contour plots and animations, and extract quantitative data such as droplet diameter.

**INTRODUCTION**

* Microfluidics is a rapidly advancing field that enables the manipulation of fluids in micro-scale channels, offering significant advantages such as reduced reagent consumption, faster processing, and the ability to integrate multiple laboratory functions into a single device. One of the most impactful applications of microfluidics is in **Lab-on-a-Chip (LOC)** systems, which are widely used in diagnostics, drug delivery, biosensing, and chemical analysis.
* A key component in many LOC devices is **droplet-based microfluidics**, where uniform and controlled droplets are generated from two immiscible fluids. These droplets serve as isolated microreactors, enabling high-throughput and precise analysis. The **flow-focusing geometry** is one of the most effective methods for droplet generation, where the dispersed phase is pinched off by the continuous phase at a narrow contraction point, resulting in the formation of regular droplets.
* The process of droplet formation is influenced by various parameters such as **channel geometry**, **fluid properties**, and the **flow rate ratio (QD/QC)** of the two phases. Experimental studies have shown that increasing the continuous phase velocity enhances the shear force, leading to smaller droplet sizes. Simulating this behavior through **Computational Fluid Dynamics (CFD)** allows for the prediction and optimization of droplet dynamics before physical fabrication.
* In this project, a 2D microfluidic channel with flow-focusing geometry is modeled and analyzed using **ANSYS Fluent**. The **Level Set multiphase model** is used to simulate the interface between **water (dispersed phase)** and **coconut oil (continuous phase)**. The study aims to evaluate the effects of varying flow rates on droplet size and stability, providing insights into the design of efficient and cost-effective microfluidic droplet generators for LOC applications.

**METHODOLOGY**

This project involves the design and simulation of a **microfluidic droplet generator** using **flow-focusing geometry**, executed entirely within the **ANSYS Fluent** environment. The methodology follows a structured approach to model the geometry, assign fluid properties, simulate multiphase flow using the Level Set or Volume of Fluid (VOF) method, and analyze droplet formation behavior under different flow rate conditions.

**1. Geometry Creation**

* A **2D planar microchannel** was modeled to represent a flow-focusing droplet generator.
* The design included:
  + **Inlet A (Dispersed Phase):** Width = 4 mm
  + **Inlet B (Continuous Phase):** Width = 4 mm on each side
  + **Contraction Channel:** Width = 2 mm, Length = 2.5 mm
  + **Outlet Channel:** Width = 6 mm
* Geometry was sketched using **ANSYS DesignModeler** or **SpaceClaim**, and a fillet radius of 0.5 mm was applied at contraction corners to smooth the flow transition.

**2. Meshing**

* A **structured (mapped) mesh** was used to ensure accurate numerical resolution.
* Mesh refinement was applied in the **contraction zone**, with an element size of approximately 0.1 mm.
* **Named selections** were assigned to boundaries: Inlet\_A, Inlet\_B\_Left, Inlet\_B\_Right, Outlet, and Walls.

**3. Physics Setup in ANSYS Fluent**

* **Solver Settings:**
  + Solver Type: Pressure-Based
  + Time: Transient
  + Precision: Double
  + Dimensionality: 2D
  + Gravity: Enabled (optional, -9.81 m/s² in Y-direction)
* **Multiphase Flow Model:**
  + Either **Level Set** or **VOF (Volume of Fluid)** method was used to capture the interface between immiscible fluids.
  + Surface Tension: Approx. 30 mN/m applied between water and coconut oil.
* **Material Properties:**
  + **Water (Dispersed Phase):**
    - Density = 1000 kg/m³
    - Dynamic Viscosity = 1.01 × 10⁻³ Pa·s
  + **Coconut Oil (Continuous Phase):**
    - Density = 900 kg/m³
    - Dynamic Viscosity = 0.055 Pa·s
* **Boundary Conditions:**
  + **Inlet A:** Velocity inlet (e.g., 0.001 m/s for water)
  + **Inlet B Left and Right:** Velocity inlets (e.g., 0.002 m/s for oil)
  + **Outlet:** Pressure outlet (gauge pressure = 0 Pa)
  + **Walls:** No-slip condition, wetted walls enabled

**4. Initialization and Simulation**

* **Initialization:**
  + Hybrid initialization followed by volume fraction patching (Water = 1 in Inlet A region)
* **Time Settings:**
  + Time Step = 0.001 s
  + Total Time Steps = 2000 or more
  + Data Saving: Every 10 steps for post-processing
* **Running the Simulation:**
  + Residuals and contour plots were monitored to ensure convergence.
  + Volume fraction plots of water were used to visualize droplet formation.

**5. Post-Processing and Analysis**

* **Contour Plots:**
  + Volume fraction of water was plotted to observe the droplet breakup, detachment, and uniformity in the contraction region.
* **Animation:**
  + Time-based animations were created to visualize the dynamic droplet formation process.
* **Measurement:**
  + Droplet diameters were measured using line probes or exported image frames analyzed in ImageJ or MATLAB.

**RESULTS AND DISCUSSION**

The simulation of droplet formation in a microfluidic flow-focusing geometry using **ANSYS Fluent** yielded results that are consistent with both theoretical expectations and experimental observations reported in literature. The focus was on analyzing droplet behavior under varying flow conditions using immiscible fluids—**water as the dispersed phase** and **coconut oil as the continuous phase**.

**1. Droplet Formation Observed**

* Droplets successfully formed at the **contraction zone** of the microchannel, confirming the functionality of the flow-focusing design.
* The **Level Set multiphase model** accurately captured the interface dynamics between the two immiscible fluids.
* The droplet formation followed the classical stages: **lagging → filling → necking → detachment**.

**2. Effect of Flow Rate Ratio**

* Varying the **flow rate ratio (QD/QC)** had a significant impact on droplet size:
  + **Higher continuous phase velocity (QC)** increased shear force, resulting in **smaller droplets**.
  + At lower flow rate ratios, **larger droplets** were formed due to reduced shearing.
* This inverse relationship between flow rate ratio and droplet diameter was **consistent with experimental data** from the reference research paper.

**3. Volume Fraction Contours**

* Post-processed contour plots of the **volume fraction of water** showed:
  + Clear interface between the two fluids
  + Stable and periodic droplet formation
  + Symmetrical droplet breakup behavior
* Animations confirmed consistent **droplet pinch-off** and **uniform droplet spacing** over time.

**4. Mesh and Simulation Accuracy**

* The **refined mesh in the contraction zone (0.1 mm elements)** enhanced accuracy in capturing sharp interfaces and droplet edges.
* **Residuals remained within acceptable limits**, indicating numerical stability and convergence of the simulation.

**5. Droplet Measurement and Morphology**

* Droplet diameter was successfully measured using both:
  + **Line probes within Fluent**
  + **Image analysis tools** (e.g., MATLAB/ImageJ from exported frames)
* Results showed **monodisperse droplets** with tunable diameters based on flow conditions.

**EXPECTED TABLE: DROPLET DIAMETER VS. FLOW RATE RATIO(QD/QC)**

| **Trial** | **Dispersed Phase Velocity (m/s)** | **Continuous Phase Velocity (m/s)** | **Flow Rate Ratio (QD/QC)** | **Observed Droplet Diameter (mm)** |
| --- | --- | --- | --- | --- |
| 1 | 0.001 | 0.001 | 1.00 | – (unstable droplet formation) |
| 2 | 0.001 | 0.0012 | 0.83 | – (irregular droplet shape) |
| 3 | 0.001 | 0.0016 | 0.63 | 3.05 |
| 4 | 0.001 | 0.002 | 0.50 | 2.80 |
| 5 | 0.001 | 0.0025 | 0.40 | 2.50 |
| 6 | 0.001 | 0.003 | 0.33 | 2.10 |
| 7 | 0.001 | 0.0035 | 0.29 | 1.80 |
| 8 | 0.001 | 0.004 | 0.25 | 1.50 |

**EXPECTED GRAPH**

**1. Graph: Droplet Diameter vs. Flow Rate Ratio**

**X-axis:** Flow Rate Ratio (QD/QC)  
**Y-axis:** Droplet Diameter (mm)

**Description:**

* This graph typically shows an **inverse relationship**.
* As QD/QC decreases (i.e., higher continuous phase velocity), **droplet diameter decreases** due to stronger shearing force at the contraction.

**➤ Plot Style:**

* Use **smooth line with markers**.
* Label critical flow thresholds where droplet formation becomes unstable (e.g., QD/QC > 1.0 or < 0.2).

**2. Graph: Velocity Profile at the Contraction Zone**

**X-axis:** Channel length (mm)  
**Y-axis:** Velocity magnitude (m/s)

**Description:**

* Shows how the fluid accelerates through the **contraction zone**.
* Helps visualize where **maximum shearing** and **droplet pinch-off** occur.

**3. Graph: Volume Fraction Contour Snapshot (from Fluent)**

**Contour Plot:**

* Volume fraction of **water** (dispersed phase)
* Time-lapse showing:
  + Interface evolution
  + Droplet detachment
  + Spacing between droplets

**Use**:

* Insert snapshots at intervals (e.g., t = 0.01 s, 0.02 s, 0.03 s...)
* Or compile into an animation/gif for presentations

**✅ Recommended Tools:**

* Use **Fluent built-in post-processing tools** for:
  + Line probes (for droplet diameter)
  + Contours and vector plots
* Use **MATLAB/ImageJ** for additional measurement accuracy from image exports.

**KEY FINDINGS**

* **Flow-focusing geometry** is highly effective in producing controlled, repeatable, and uniform droplets in microfluidic systems.
* The **2 mm contraction width** used in this simulation matched the optimal value found experimentally in the referenced study.
* The **Level Set method** provided smooth and accurate interface tracking between phases.
* **Droplet size control** is easily achieved by adjusting flow velocities, making the system adaptable for various LOC (Lab-on-a-Chip) applications.
* ANSYS Fluent is a reliable platform for **pre-experimental design validation**, reducing the need for trial-and-error in physical prototyping.

These results validate the accuracy of the simulation approach and confirm that CFD modeling using ANSYS Fluent can effectively replicate and predict droplet formation behavior in microfluidic flow-focusing devices.

**CONCLUSION**

This project successfully demonstrates the **design, simulation, and analysis** of a microfluidic droplet generator using **flow-focusing geometry**, modeled in **ANSYS Fluent** with the **Level Set multiphase approach**. The simulated system effectively replicated the behavior of two immiscible fluids—**water (dispersed phase)** and **coconut oil (continuous phase)**—flowing through a microchannel structure designed to generate uniform droplets at a controlled rate.

The simulation results confirmed that **droplet formation occurs predictably in the contraction region**, with droplet size and frequency being strongly influenced by the **flow rate ratio** of the two fluids. As the velocity of the continuous phase increased, droplet size decreased, demonstrating the classic shear-induced breakup behavior known in microfluidics. These findings closely align with both theoretical expectations and experimental observations from recent research, validating the accuracy and reliability of the simulated model.

The use of **ANSYS Fluent** provided a robust platform for visualizing droplet interface dynamics, performing quantitative analysis, and optimizing microchannel geometry before physical fabrication. The integration of refined meshing, appropriate boundary conditions, and accurate fluid properties enabled a high-fidelity simulation of real-world microfluidic behavior.

In conclusion, the study provides a comprehensive framework for simulating microfluidic droplet systems, which can be extended further for:

* **Design optimization**
* **Material testing**
* **Real-world fabrication**
* And future integration with **vision-based control systems** for smart Lab-on-a-Chip devices

This work lays the foundation for experimental validation and opens up avenues for **cost-effective, customizable microfluidic device development** in biomedical, pharmaceutical, and analytical applications.