

I. INTRODUCTION

Fluid flow in a spatial domain having the length of atleast one of its spatial dimension of the order of microns is termed as microfluid flow. Devices employing microfluid flow for their function are termed as microdevices.

Microdevices are useful because they allow manipulation with fast response times, they can handle small fluid volumes, sense and control flows and pattern substrates on small length scales, and, what promises to be very important, they can selectively address the cellular scale [1].

Microfabricated integrated circuits revolutionized computation by vastly reducing the space, labour, and time required for calculations. Microfluidic device systems hold similar promise for the large scale automation of chemistry and biology, suggesting the possibility of numerous experiments performed rapidly and in parallel, while consuming little reagent [2].

The design of such microfluidic device systems involves scientific understanding of the flow of the fluids used in the devices and of their interaction with the structure of the device. This leads to a mathematical model of the microfluidic system based on scientific principles. Such models help us mathematically represent facets of device function such as the volume throughput in case of microfluidic mixers or separation efficiency in case of filters etc. These facets of device function are related to properties of the fluid, device structure and input parameters (like applied pressure difference or electric fields) This helps designers to simulate the functioning of the device system using a mathematical model and predict various aspects of the performance of the design.

II. A MODELING EXERCISE

When one designs microfluidic devices, one encounters practical issues of engineering wherein empirical or phenomenological rules, experience of practitioners and rules of thumb often serve as guiding principles. I describe below one such instance where I was interacting with a device fabricator.

The fabricator had some flow data which contained variation of volume flow rate with pressure differences across the serpentine channel that was being used. The channel had rough walls and we attempted some phenomenology.

The Colebrook Formula told us about how the friction of the channel walls related to the Reynolds number of the flow.

$$\frac{1}{\sqrt{(\lambda)}} = 1.74 - 2 \times \frac{LOG}{10} \left(\frac{k}{R} + \frac{18.7}{Re\sqrt{(\lambda)}} \right) \quad (2.1)$$

where λ is the friction factor, k is the height of average protrusion inside the channel, R is the average diameter of the channel and Re is the Reynolds number.

The formula above is valid only for straight channels. We first found out the equivalent length of the serpentine channels by fitting this formula to data. This formula, we later realized, is for the turbulent flow regime. However, the data we had implied $Re < 2000$. So, we simply found the equivalent length by fitting Hagen Poisseulle formula

$$\dot{V} = \frac{\pi P a^4}{8\eta L} \quad (2.2)$$

where \dot{V} is the volume flow rate, a is the average radius of channel

$C/S, P$ is the pressure difference and L is the equivalent length.

I then promised to try and work out a formula for a single circular 180 degree bent portion and we planned to piece together the pressure differences.

This was a simple attempt at modeling the microfluidic device, though not very sophisticated. We managed to obtain a validation to within 10 percent of experimental observations though it worked fine only for low Reynolds numbers.

The above model of microfluidics suffered from several drawbacks. Firstly, it was limited to low Reynolds numbers and to those serpentine channels which have circular bends. Secondly, the model is limited by the design requirements of serpentine channels. Microfluidic systems are much more diverse in terms of structure and function than a single

device like a particular channel. Perhaps, the most prominent shortcoming that appears in the above exercise is the lack of consideration of complexities that exist in microfluidic flows. A considerable body of knowledge has emerged regarding the science of microfluidics and its relation to aspects of device technology which we outline in the next section.

III. MICROFLUIDIC SYSTEMS: OVERVIEW AND SCIENTIFIC BACKGROUND

Microfluidic devices are of many types like reservoirs, heat exchangers etc. and we briefly mention the generic structure of a microfluidic system and outline design considerations.

A microfluidic system must have a series of generic components [3]:

- 1) A facility for introducing reagents and samples (probably as fluids, although ideally with the option to use powders);
- 2) Facilities for moving these fluids around on the chip, and for combining and mixing them;
- 3) Various other devices (such as detectors for most microanalytical work, and components for purification of products for systems used in synthesis)

The field has, so far, centred on demonstrating concepts and models for these components i.e. microfluidic devices individually rather than the entire microfluidic system.

There are many types of microfluidic devices which I classify on the basis of function as (a more comprehensive list can be found online in "Microfluid Mechanics" on the Wikiversity website)

- 1) Pumps (eg. pressure driven, electroosmotic)
- 2) Valves
- 3) Actuators
- 4) Sensors
- 5) Mixers
- 6) Reactors
- 7) Purifiers
- 8) Channels
- 9) Reservoirs

There seem to be five distinct physical structural factors which influence the design of these microfluidic devices. 1) Geometry of device

- 2) Surface structure and properties (surface tension,electrical effects, van der Waals interactions and surface roughness)
- 3) Subsurface structure and properties
- 4) Additional micro, nano and sub nano structures (eg dispersed phase droplets or bubbles, suspended particles, Angstrom sized organic molecules)
- 5) Wall elasticity and active fluid control elements (like hydrogels)

In addition to structural factors, the physics of small volumes(micro- nano and pico- litres) of the fluids is crucial for design considerations. This is termed as microhydrodynamics in literature. It could be parametrized by a series of dimensionless numbers expressing the relative importance of various physical phenomena [2]. These numbers are:

Reynolds number Re , addressing inertial effects

Peclet number Pe , which concerns convective and diffusive transport

Capillary number Ca expressing the importance of interfacial tension

Deborah, Weissenberg, and Elasticity numbers De , Wi , and El , describing elastic effects due to deformable microstructural elements like polymers in the fluid

Grashof and Rayleigh numbers Gr and Ra , describing density-driven flows on account of gravity

Knudsen number Kn , describing the importance of noncontinuum molecular effects. Depending upon the relative importance of these numbers, specific scientific models are constructed for the physical phenomenon and mathematical expressions for dynamic quantities like forces, mobilities, flow rates etc. are sought to be derived. It would be worthwhile to distinguish between various kinds of forces that are involved so that separate scientific models could be constructed for them. The prominent physical forces which turn out to be important in microhydrodynamics can be broadly classified as

- 1) Surface Tension
- 2) Viscosity
- 3) Electrical effects due to double layers
- 4) van der Waals forces

5) Force due to surface roughness

These effects are tailored by engineering at various levels of structure described above, for designing the functional device.

There are thus two facets of the scientific model of a microfluidic device, one concerning the structure function relationship in the fluid (including additional particles which may be inside it) and the other concerning the structure function relationship of the solid surrounding the fluid vis-a-vis its interaction with the fluid medium it contains. I briefly described the former before. I outline the latter below.

Owing to the small dimensions of microfluidic devices and to the long-range nature of viscous flows, boundaries of the fluid domain are never very far in microfluidics. Therefore, techniques that exploit boundary effects emanating from the surrounding solid medium can be quite effective in microfluidic manipulation. Three important phenomena which relate to this are electrokinetic effects which involve electric charge density on the device walls bounding the fluid, acoustic streaming wherein high frequency sound waves are used to effect fluid displacement along channel lengths which again involves boundaries, and fluid-structure interactions wherein the mechanical displacement of surrounding solid of the device is used to drive the fluid. Apart from the pump devices which force the fluid by applied pressure differences or by electrokinetic effects, microfluids are also manipulated by

- 1) Capillary driving forces owing to wetting of surface by fluid
- 2) Free surface flows due to gradients of interfacial tension (Marangoni Flows)

The mathematical models for these effects are similar to the force based models we construct for the fluids.

Thus the model of a device has to have both the features of the fluid as well as those of the surrounding device wall incorporated in it based upon the scientific understanding outlined above.

IV. MATHEMATICAL MODELING

The behaviour of fluid and its interaction with its surroundings can be modeled at four levels:

- 1) Continuum Scale
- 2) Meso scale
- 3) Micro scale (atomic)
- 4) Sub-micro Scale (sub-atomic)

The fluid flow in devices is thus modeled incorporating the device structural factors appearing as mostly side conditions on the differential equations involved and the constitutive laws of the fluids which depend largely on the fluid chemical structure and fluid nano-level structure.

For microscopic liquid and multiphase flows, although the fluid can be still described as a continuum substance, due to the high surface-area-to-volume ratio, microscopic interactions and related interfacial phenomena, such as electrokinetics, surface wettability, and interfacial slip, which are typically neglected in macroscopic systems, have to be considered carefully [4].

Some discrepancies have been reported between flow measurements made in small channels and expectations from classical theory based on solutions to the Navier-Stokes equations. However, a careful consideration of the experimental results for pressure-driven flows of liquids demonstrates that, in fact, in almost all cases there are no significant discrepancies [1].

Thus, it is our understanding that for small molecule liquids such as water, the familiar continuum description remains an appropriate starting point for analysis of microdevices, with appropriate consideration being given to electrical effects, slip, etc.

A. Continuum Modeling

At this level of modeling matter, the fluid is regarded as a continuum, and the well-known macroscopic laws (equations of continuity and momentum conservation, constitutive equations such as Fourier's law, etc.) apply [5].

An important feature in such models of microfluids is the occurrence of multiphase fluids. The conservation and constitutive laws are therefore accordingly considered. They have an additional set of dependent variables called phase volume fractions [6].

The balance of mass for each phase is

$$\frac{\partial \alpha_N \rho_N}{\partial t} + \nabla \cdot \alpha_N \rho_N \vec{v}_N = S_N \quad (4.1)$$

where N is a label for the phase
 α_N is the phase volume fraction
 ρ_N is the density of the particular phase
 \vec{v}_N is the velocity of the N phase particle
 S_N is the time rate transfer of mass to phase N per unit volume

As the overall mass is conserved we have

$$\sum_{N=1}^{\text{no.of phases}} S_N = 0 \quad (4.2)$$

The momentum balance equation turns out to be

$$\frac{\partial \alpha_N \rho_N \vec{v}_N}{\partial t} + \nabla \cdot \left(-\overleftrightarrow{\tau}_N + \alpha_N \rho_N \vec{v}_N \vec{v}_N \right) = \alpha_N \rho_N \vec{K} + \vec{F}_N \quad (4.3)$$

where $\overleftrightarrow{\tau}_N$ is the stress tensor for phase N

(various dimensionless numbers mentioned above play an important role in setting up relevant constitutive laws for the stress tensor)

\vec{K} is the bulk force per unit mass

\vec{F}_N is the momentum transferred to the phase N from the other phases per unit volume per unit time. It is also called "force interaction".

As the overall phase interaction forces cancel out, we have

$$\sum_{N=1}^{\text{no.of phases}} \vec{F}_N = \vec{0} \quad (4.4)$$

Similarly energy balance leads to

$$\frac{\partial (\alpha_N \rho_N e_N + 1/2 \alpha_N \rho_N |\vec{v}_N|^2)}{\partial t} + \nabla \cdot \left((\alpha_N \rho_N e_N + 1/2 \alpha_N \rho_N |\vec{v}_N|^2) \vec{v}_N + \overleftrightarrow{\tau}_N \cdot \vec{v}_N - \vec{Q}_N \right) = \rho_N \vec{v}_N \cdot \vec{K} + W_N \quad (4.5)$$

where W_N is the energy transferred to the phase N fluid from other phases per unit volume per unit time.

\vec{Q}_N is the heat flux density within the phase N fluid

e_N is the internal energy per unit mass corresponding to the phase N fluid.

Separate dynamics is required to be set up for the volume fractions α_N that appear in the above equations. In the context of microfluidics, this appears as a (set of) scalar fields which are ultimately employed to track interfaces between phases that flow in the microfluidic device. Various models are employed and they could be broadly classified into two types, one leading to a sharp (infinitely thin) interface and the other type leading to a diffuse interface [5]. Much of the difference in the mathematical models takes place at this stage. These set of partial differential equations are numerically represented as difference equations and these are further solved employing computational algorithms. Several issues arise at the computational level ranging from computational effort to accuracy and stability of the approximate solutions obtained. I shall not address such issues here but demonstrate an example later of how differential equations help us obtain design insights.

B. Meso Scale Modeling: Lattice Boltzmann method

Meso Scale modeling involves the the mesoscale level, in which matter is regarded as composed of blobs of matter, each containing a number of atoms [5] At this scale thermodynamic fluctuations are seen to prominently affect physical results. One could say that such flows fall in the regime of Knudsen numbers beyond 0.2 . Lattice Boltzmann (LB) method is a meso scale method of modeling.

This method has been applied to microfluidic flows in various contexts [4]

- 1)Magnetohydrodynamic flows
- 2)Electrokinetic flows and electrohydrodynamic droplet deformation
- 3)Microflows in porous media
- 4)Biological microfluidics
- 5)Solid Liquid interfacial slip

6) Multiphase flows and surface effects

7) Microscopic gaseous flows

LB approach is a simple explicit algorithm which can be derived from the linearized Boltzmann equation and is often associated with a square (in 2D) or cubic (in 3D) lattice on which the discretized particle distribution function f evolves. The velocities of the particles can take only discrete values (this set of values is the same under sign reversal) The particles of the distribution move synchronously from a grid point to its neighbours according to the velocities they possess. The difference equation for f is taken to be

$$f_A(\vec{r}_i + \vec{c}_A \delta t, t + \delta t) = f_A(\vec{r}_i, t) + \delta t (\Omega_A(f(\vec{r}_i, t)) + F_A(\vec{r}_i, t)) \quad (4.6)$$

where Ω is called the collision term and F is the external forcing. i is an index that runs over cell labels A is an index which runs over the labels of the discrete velocities that have been stipulated in the LB model

An LB time step involves two substeps

- (i) a streaming substep, in which particles (in the distribution function) move to their neighboring sites
- (ii) a collision substep, where these particles collide with particles coming in from the other directions. The distribution functions are changed by the effects of interparticle collisions (which may for instance account for intermolecular forces)

Macroscopic quantities such as the density and the fluid velocity are obtained by taking suitable moments of the distribution function

$$\rho = \sum_{A=0}^Q f_A \quad (4.7)$$

$$\vec{u} = 1/\rho \sum_{A=0}^Q f_A \vec{c}_A \quad (4.8)$$

where ρ is the density

\vec{c}_A is the discrete velocity labelled by A .

Two important issues occur in LB methods involving multiphase flows, one concerns representation of the multiple phases and the other concerns simplifying the collision term Ω above. In multiphase flows one usually either employs as many sets of populations as phases or the free energy approach having populations again but describing fluid fractions [5].

The collision integral is usually simplified by expressing it as a relaxation term with a time scale τ towards a suitable equilibrium. Multiple relaxation times are employed when one has multiphase flows.

1. Advantages and Disadvantages

The numerical method is easier to implement as compared to continuum methods as Boltzmann equations are semilinear. Complex geometries can be handled in a simpler manner and there is better scope for parallelization of the algorithms. LB methods are found to maintain interfaces with lesser computational effort than continuum methods.

In LB methods intensive macroscopic fluid properties and macroscopic parameters cannot be prescribed as initial conditions or parameters for the difference equations solved. Thermophysical properties need to be related to the distribution function/s before and require separate models for that. Difficulties are encountered while developing such models for boundary conditions as well both from the scientific conceptual (like mass conservation) and the computational (CPU time) angles.

C. Micro- and sub-micro scale modeling

The number of atoms or of molecules in a short channel of a square nanometre sized cross-section having a length of a few microns is typically of the order of a few hundred thousand. It is conceivable to simulate such a matter portion using molecular dynamics. The fluid flow could be said to take place at high Knudsen numbers and high Deborah numbers. Each individual atom or molecule is ascribed state variables (position, momentum and maybe

other internal degrees of freedom) which change in time depending upon states of other atoms or molecules in the matter portion. The nature of interactions between the atomic, ionic or molecular species (or agents as termed in Agent Based Models) has been understood from the scientific angle in terms of largely electrostatic interactions [7]. Such interactions are important in stipulating the time evolution of the state above. With increasing availability of higher computer processing speeds, such simulations can be carried out nowadays for sizes of devices ranging in the above range (like 100 nm. and 100,000 atoms or agents) [8],[9]. It would certainly be a some years before such "direct numerical simulations" could address a device which is a few millimetre in length.

Scientific modeling at a scale near or below the atomic involves quantum physics and simulations based on this are carried out using a model representing the electron density popular in the chemistry community, called the density functional theory. Simulations at this scale are expensive in terms of computing effort. We can currently implement simulations for upto a few hundred atoms or molecules. It would be much longer before we could expect sub-micro scale simulations to be carried out which would be useful for microfluidic applications.

V. CONTINUUM MODELING: EXAMPLE OF APPLICATION

Amongst the several types of devices, let us consider the mixer in which two or more phases of fluid are mixed to form a dispersion or even a solution. There are two types of mixers, the active ones in which external energy (other than the energy in the fluid) is provided to the device to facilitate the mixing, and the passive ones wherein only the energy of the fluid is used for mixing coupled with device geometry and surface conditions.

I describe the details of the design below and then show how continuum modeling has been carried out to yield a design insight.

A. A design of a passive micromixer

An example of modeling and fabrication of a 3D micromixer based on the principle of sequential lamination is provided in [10].

The design approach is supported by analytical treatment which indicates superiority of 2D transverse flows while designing a micromixer. We show in this section how mathematical modeling is carried out at the core of this analytical treatment. The model relies on the application of partial differential equations and the analysis employs properties of its mathematical solutions.

Before we describe the model, a brief description and picture of the passive micromixer is provided below.

The micromixer is designed on the principle of sequential lamination [11]. For the initial design, a small channel *a* was added to the node when the fluid splits into the side channels *b*. This heavily altered the behavior of the mixer thus formed. A reason for this behavior can be understood by considering the velocity profile of fluid flow at the node where channel *b* originates (Fig 2). It is seen that the fluid recombining at the node generates certain amount of mixed fluid due to increased diffusion by convective force, in direction perpendicular to length axis. Most of this mixed fluid is extracted by the channel *b* and thus the nature of the fluid that recombines in the next node is almost similar to the previous node.

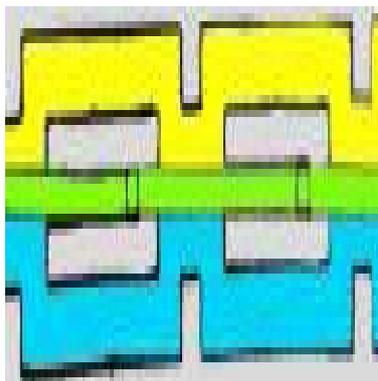


FIG. 1: :A picture of the microchannel system (Top View)

A CAD diagram of the micromixer is shown in the figure above. All dimensions are in microns. The geometry studied extensively was with the side channels being rectangular. The two inlet channels meet into the channel *d* which

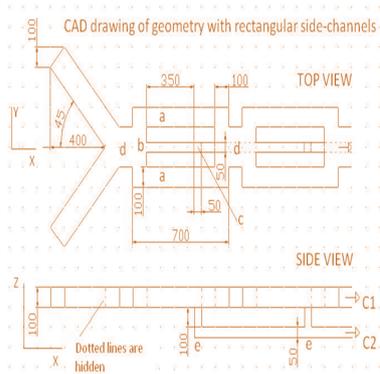


FIG. 2: :A CAD drawing of the micromixer portion

splits into three channels, two side channels *a* and a smaller channel *b* between the side channels, which recombines along with the side channels again before the next similar unit begins. One such unit is called a node. More such nodes or units can be repeated, as indicated by the arrow. The entire arrangement is called channel C1. The channel *b* turns downwards sending the fluid into the hole, denoted as *c*, through which the fluid enters a channel C2 below, via the elbow *e* (indicated in the side view). The region where the fluid is extracted is important and is modeled using continuum mechanics below. The rest of the details of the design are regarding the outlets. I do not address these in the model, but will provide them here for completeness. The cross section of the channel C2 is $50 \times 50 \mu\text{m}$. The mixer terminates with two outlets, one bigger outlet (called main-outlet (O1) in further discussions) at the end of the last node, in C1, and the other smaller outlet (called mini-outlet, or O2) at the end of C2. The region where the fluid is extracted is important in the model as we find that it illustrates how monotonic concentration cross sectional profiles could be obtained as against non monotonic ones.

B. Modeling Channel C1 near the extraction outlet

We approximate the flow in the channel near the end where the mixture is being extracted. The flow is assumed to be largely within the transverse cross-section of the channel in this region and so we assume a two dimensional flow model. The fluid mixture has concentration of ethanol c and of water $1 - c$. This is similar to the volume phase fraction α in multiphase flows. We apply the continuity equation

$$\frac{\partial c}{\partial t} = -\nabla \cdot (c\vec{v}) + D\nabla^2 c \quad (5.1)$$

where $c(x, y)$ is the concentration profile of ethanol with respect to transverse Cartesian coordinates x and y . \vec{v} is the velocity in the transverse plane. D is the diffusivity of ethanol in water. This is treated as a form of application of multiphase equations for two phases which mix with each other (ethanol and water) If we assume a steady state, then we have a potential flow as a possible solution

$$\vec{v} = \nabla(D \ln c) \quad (5.2)$$

We take this solution and apply the momentum balance equation in the transverse plane. We assume that there is no pressure gradient in the transverse direction

$$\vec{v} \cdot \nabla \vec{v} = \frac{\eta/3 + \lambda^*}{\rho} \nabla (\nabla \cdot \vec{v}) + \frac{\eta}{\rho} \nabla^2 \vec{v} \quad (5.3)$$

where ρ is the density, η is the dynamic viscosity coefficient and λ^* is the bulk viscosity coefficient.

The above two equations yields a set of two partial differential equations in Cartesian co-ordinates. They become particularly easy to integrate once if ρ had been constant. But the same simplification can be utilized in a slightly different manner below. Assume that ρ is dependent on concentration c only and can be series expanded as

$$\rho = \rho_0 + \rho_1 c^1 + \rho_2 c^2 + \dots \quad (5.4)$$

This is substituted in the nonlinear differential equation and retaining the leading order contribution to ρ , one obtains

$$\left(\frac{\partial \ln c}{\partial x}\right)^2 + \left(\frac{\partial \ln c}{\partial y}\right)^2 = \frac{-2K}{\rho_0 D} \left(\frac{\partial^2 \ln c}{\partial x^2} + \frac{\partial^2 \ln c}{\partial y^2}\right) \quad (5.5)$$

where $K = \frac{4\eta}{3} + \lambda^*$

This nonlinear differential equation can be re-cast by multiplying both sides by $c^{\frac{\rho}{2\eta}}$ and some algebra to the form

$$\frac{\partial^2(c^{-\frac{\rho_0 D}{2K}})}{\partial x^2} + \frac{\partial^2(c^{-\frac{\rho_0 D}{2K}})}{\partial y^2} = 0 \quad (5.6)$$

which is the Laplace equation in two dimensions.

We could assume suitable boundary conditions for the concentration c . For example, c is 0 on the left half of the square boundary and 1 on the right half of the boundary in keeping with the fact that the left channel carries water and the right carries ethanol. No slip condition is automatically guaranteed here since c does not change along the wall. However the potential flow solutions like the one we propose usually encounter difficulties at boundaries and a boundary layer solution is resorted to near these regions [12]. In the interior, the potential flow solution would be realistic and the above Laplace equation could be interpreted there. From the mathematical properties of solutions to Laplace equations, we infer that c cannot possess any extrema in the interior. This forces monotonic behavior of concentration and thus any mixing that occurs would tend to be uniform as against any clumped behavior.

Instead of modeling the region with a transverse barrier where the mixture is extracted, we now address the three dimensional channel where there is no barrier. One applies the same procedure as in the previous section for solving equations of continuity, potential flow and momentum balance three dimensions. The channel is modeled as a cuboid. The X and Y directions are the transverse directions in the cross-sectional plane of the channel. The Z direction is the centre line which is along the length of the channel. Manipulations similar to the ones in the previous section can be carried out in three dimensions

$$\left(\frac{\partial \ln c}{\partial x}\right)^2 + \left(\frac{\partial \ln c}{\partial y}\right)^2 + \left(\frac{\partial \ln c}{\partial z}\right)^2 = \frac{2K}{\rho_0 D} \left(\frac{\partial^2 \ln c}{\partial x^2} + \frac{\partial^2 \ln c}{\partial y^2} + \frac{\partial^2 \ln c}{\partial z^2}\right) - \frac{P}{\rho_0 D^2} \quad (5.7)$$

where P is the pressure. This leads further by steps similar to the previous section to the three dimensional equation

$$\frac{\partial^2(c^{-\frac{\rho_0 D}{2K}})}{\partial x^2} + \frac{\partial^2(c^{-\frac{\rho_0 D}{2K}})}{\partial y^2} + \frac{\partial^2(c^{-\frac{\rho_0 D}{2K}})}{\partial z^2} + \frac{c^{-\frac{\rho_0 D}{2K}} P}{\rho_0 D^2 K^2} = 0 \quad (5.8)$$

We separate the transverse and axial parts using the method of separation of variables to obtain the Helmholtz equation (for the transverse part $c_T(x, y)$, where $c(x, y, z) = C_T(x, y)Z(z)$)

$$\frac{\partial^2(c_T^{-\frac{\rho_0 D}{2K}})}{\partial x^2} + \frac{\partial^2(c_T^{-\frac{\rho_0 D}{2K}})}{\partial y^2} + b c_T^{-\frac{\rho_0 D}{2K}} = 0 \quad (5.9)$$

where b is a constant. The axial part can be obtained exactly in terms of hyperbolic or exponential functions of z .

It is important to note that the three dimensional equation above will simply reproduce the solution of the two dimensional Laplace equation if we demand that the axial velocity is zero (or axial pressure gradients are negligible) as we have in the previous section. However, if that is not the case (and we choose to study the transverse velocity components only) then we would need to compare with a two dimensional Helmholtz equation assuming we use separation of variables for the z part. Such a Helmholtz equation would yield (positive) solutions for the concentration which allow for extrema and hence clumping in the transverse direction unlike the monotonically behaving transverse profile of the Laplace equation. This observation motivates us to modify the channel so that there are more regions which can be modeled by the Laplacian equation. It is suggested that monotonic behavior is an indication that the clumped mixing is not allowed. If one is able to ensure that the axial velocity almost vanishes in some parts of the cross section, then one would expect more uniform mixing.

Thus by way of comparing solutions of two differential equations, we are able to suggest that concentration profiles can be expected to be monotonic across the channel cross section when the axial speeds are small. This supports designing channels where transverse fluid motion is enhanced whenever such profiles are desired.

VI. CONCLUSION

Microfluidic device systems hold promise for efficient technology in the future and designing these systems would carry great importance. As an aid to design, one develops mathematical models which help simulate the behaviour of the devices in the system. I have surveyed and summarized various scientific principles involved, provided a general picture of microfluidic device systems and outlined different levels at which models can be constructed for their component devices. Finally, I have demonstrated an example in which continuum modeling has been carried out for a particular aspect of a micromixer.

-
- [1] H. A. Stone, S. Kim *AIChE Journal* **47** (2001) 12501
 - [2] Todd M. Squires, Stephen R. Quake *Rev. Mod. Phys.* **77** 977 (2005)
 - [3] George M. Whitesides *Nature* **442** (2006) 368
 - [4] Junfeng Zhang *Microfluid Nanofluid* **10** (2011)
 - [5] Martin Woerner *Microfluid Nanofluid* **12** (2012) 841
 - [6] *Fundamentals of Multiphase Flows* Christopher E. Brennen (2005) Cambridge University Press
 - [7] Israelachvili, J.N. *Intermolecular and Surface Forces* Academic Press, London, 2011.
 - [8] Shaurya Prakash, Junghoon Yeom *Nanofluidics and Microfluidics: Systems and Applications* (2014)
 - [9] Suvankar Ganguly *Particle Transport in Nanoscale Colloidal Suspensions in Microfluidics and Microscale Transport Processes* (2012) 265 ed. Suman Chakraborty, CRC Press
 - [10] Suresh Gosavi, Aniket Tekawade, Dhananjay Bodas, Sukratu Barve, Laurent Robert, Chantal Khan-Malek *Proceedings of the American Society for Mechanical Engineers*, 9th International Conference on Nanochannels, Microchannels and Minichannels, 2011, Edmonton, Canada.
 - [11] N.T. Nguyen, *Fundamentals and Applications of Microfluidics* (2006)
 - [12] B. J. Kirby, *Micro-and Nano scale Fluid Mechanics*, Cambridge University press, Potential fluid flows, Chapter 7, pp. 156 NY (2010)