Design and Analysis of Micro-Fluidic Channel of Different Configurations

Basavaraj Indi

Department of Mechanical Engineering, SVERI's College of Engineering,

Final Year Engineering Student

INTRODUCTION:

Microfluidics is the study of fluid flow in geometries with one of the channel dimensions being of the micrometer scale. These geometries are built-up into circuits known as microfluidic chips. This technology has been the cause for much research, as it provides a means for carrying out key chemical assessment processes in the biomedical field. This technique has advantages over the standard bench-top method due to the low volume of reagents needed and the higher speed of the analysis. Other advantages include the fact that they are readily automated, parallelizable, and portable and have relatively low material cost. This technology has many applications in many different fields including pharmaceuticals, cosmetics, medicine and biotechnology. Main applications of micro fluids at this stage include diagnostics, DNA sequencing, drug delivery, lab-on-a-chip applications, micro-reactors, and fuel cells. One of the main challenges in microchannel is mixing where more than one fluids come together. It is difficult to get a uniform mixing in Microsystems due to the laminar nature of the most micro flows. Various techniques enhance fluid micro mixing and their application for biological and chemical processes. One important application of the microfluidic devices is for biological processing where rapid mixing is usually an important step. Besides biological analysis, another application field of the fluid micro mixing technology is in micro reactor which may bring revolutionary influence on modern chemistry. Micro-reactors will be a good choice for chemists working with very expensive materials or materials only available in small amounts. The mixing efficiency of the reactants is mainly influenced by the reaction time, the yield as well as the quality of the final result. Other parameters in micro-scale flow forces such as tension, interfacial tension and Van der Waal's molecular forces are more apparent than in macro-scale flow.

The shape of microchannels is an important design variable to achieve the desired performance. Since most microchannels are, at present, designed by trial and error, a systematic shape design method needs to be established. Computational fluid dynamics (CFD) is often used to rigorously examine the influence of the shape of microchannels on heat and mass transport phenomena in the flow field. Recently, the adjoints variable method has attracted the attention as an efficient sensitivity analysis method, particularly for aeronautical shape design, since it allows one to successfully obtain the shape gradient functions independently of the number of design variables.

Large surface to volume ratio of cross-section and high aspect ratio of length to cross-section are two key geometric characteristics of microchannels. Consequently, the boundary layer takes a greater volumetric share in microchannels than in larger macroscopic channels. The quality of the channel surface and rate of change in the cross-section of the microchannel will have a strong effect on flow behavior. With an increase of the ratios of (i) wall surface to flow volume and (ii) channel length to cross-section, the resistance to flow in a microchannel becomes high, thus requiring high pressure to drive the flow through the device. This is a typical behaviour for microchannels which tend to have a high resistance to flow resulting in low flow velocities. Geometric features of microchannels result in high computational demands for Computation Fluid Dynamics due to the need for a high resolution (i.e. fine) mesh. A fine mesh is required near the channel walls to adequately capture the boundary layer effects. To maintain suitable mesh aspect ratios, with respect to the axial location, requires an extremely large number of computational cells.

The simulations are used to guide tradeoffs between heat transfer performance and requirements of the fabrication process. In addition to fluid flow, the models predict the solid material thermal fields. Local thermal gradients cause local strain through thermal expansion, which in turn can lead to failure of the ceramics. The current design is in its fifth generation, with each generation improving performance and manufacturability. The design is based upon simulation using three-dimensional computational fluid dynamics (CFD), including the conjugate heat transfer between fluids and solid materials. The CFD models are implemented in Fluent. Temperature-dependent properties are used for air and for alumina.

Because the field of microfluidics is a relatively immature field, numerical simulations of microfluidic systems can be extremely valuable both in terms of providing a research tool and as an efficient design and optimization tool. By incorporating the complexities of channel geometry, fluid flow rates, diffusion coefficients and possible chemical interactions into a numerical model, the behavior of a particular system can be accurately predicted when an intuitive prediction may be extremely difficult. For example, a custom coded numerical model has been used to predict differences in the diffusive scaling laws across the depth of a micro channel. Numerical modeling also allows visualization of complex flow phenomena that may not be easily obtained experimentally.

However, perhaps the greatest use of numerical modeling, particular in the form of a convenient commercially available package, is as a design tool. In this case, the model can be used to explore the effects of changing any number of parameters on the performance of the device without actually fabricating different devices. For example, different channel geometries can be simulated to explore the extent of inter diffusion or chemical binding along the centerline of a T-Sensor to determine the parameters required for a desired diffusion profile.

The weakness of numerical modeling is, of course, the fact that it is not guaranteed to exactly replicate events in nature, particularly if there are physical phenomena that are not incorporated into the model. However, careful examination of simulation results and comparison of numerical and experimental data can validate the use of the model as a predictive tool. The commercial modeling package used in this research has been shown to be reasonably accurate, both from comparisons with experimentally obtained images and from comparison with a different numerical modeling approach. These features will be described in more detail below.

PRESENT THEORIES AND PRACTICES:

Duan et al. have demonstrated that microscale fluid dynamics has received intensive interest due to the emergence of Micro-Electro-Mechanical Systems (MEMS) technology. When the mean free path of the gas is comparable to the channel's characteristic dimension, the continuum assumption is no longer valid and a velocity slip may occur at the duct walls. The elliptic cross-section is one useful channel shape that may be produced by microfabrication. The elliptic microchannels have potential practical applications in MEMS. Slip flow in elliptic microchannels has been examined and a detailed theoretical analysis has been performed. A solution is obtained using elliptic cylinder coordinates and the separation of variables method.

Gunnasegaran et al. [2] has described the effect of geometrical parameters on water flow and heat transfer characteristics in microchannels. The three-dimensional steady, laminar flow and heat transfer governing equations are solved using finite volume method. The computational domain is taken as the entire heat sink including the inlet/outlet ports, wall plenums, and microchannels. Three different shapes of microchannel heat sinks were investigated in these studies which are rectangular, trapezoidal, and triangular. The water flow field and heat transfer phenomena inside each shape of heated microchannels are examined with three different geometrical dimensions. Using the averaged fluid temperature and heat transfer coefficient in each shape of the heat sink to quantify the fluid flow and temperature distributions, it is found that better uniformities in heat transfer coefficient and temperature can be obtained in heat sinks having the smallest hydraulic diameter. It is also inferred that the heat sink having the smallest hydraulic diameter has better performance in terms of pressure drop and friction factor among other heat sinks studied.

Hooman et al. [3] have offered theoretical results for fully developed slip-flow forced convection through a microchannel of semicircular cross-section. Numerical results are also presented to study the developing region. Velocity slip and temperature jump boundary conditions are applied at the uniformly heated walls. The results from the two different sources are cross validated and those pertaining to the limiting case of no-slip flow are found to be in good agreement with those available in the literature.

Masahiro et al. [4] have investigated hydraulic diameter experimentally and numerically. Air and water superficial velocities were 0.018–0.791 m/s and 0.042–0.757 m/s, respectively. Three-dimensional modeling was performed with computational fluid dynamics (CFD) software FLUENT and the volume of fluid (VOF) model. Slug flow (snapping/breaking/jetting) and stratified flow were observed experimentally. Numerically predicted void fraction followed a linear relationship with the homogeneous void fraction, while experimental values depended on the superficial velocity ratio UG/UL. Higher experimental velocity slip caused by gas inlet pressure build-up and oscillation caused deviation from numerical predictions. Velocity slip was found to depend on the cross-sectional area coverage of the gas slug, the formation of a liquid film and the presence of liquid at the channel corners. Numerical modeling was found to require improvement to treat the contact angle and contact line slip, and could benefit from the use of a dynamic boundary condition to simulate the compressible gas phase inlet reservoir.

Qu et al. [5] conducted experiments to investigate flow characteristics of water through trapezoidal silicon microchannels with a hydraulic diameter ranging from 51 μm to 169 μm. Their results indicate that the pressure gradient and flow friction in microchannels are higher than those given by the conventional laminar flow theory due to the effect of surface roughness of the microchannels. So, they proposed a roughness–viscosity model to interpret the experimental data.

Steinke et al. [6] presented a comprehensive review of friction factor data in microchannels with liquid flows. They indicated that entrance and exit losses need to be accounted for while presenting overall friction factor losses in microchannels. Most of the data that accounted for friction factor loss show good agreement with the conventional theory. They also provided a new procedure for correcting measured pressure drop to account for inlet and outlet exit losses.

Three-dimensional fluid flow and heat transfer phenomena inside heated microchannels were investigated by Toh et al. [7]. They solved the steady laminar flow and heat transfer equations using a finite-volume method. The numerical procedure was validated by comparing the predicted local thermal resistances and friction factor with the available experimental data. They have found that the heat input lowers the frictional losses and viscosity leading to an increase in the temperature of the water, particularly at lower Reynolds numbers.

Xue et al. [8] had studied the behaviour of a fluid, which may contain particle suspensions, flowing in micro-dimensional channels. The author discussed some of the key design factors affecting fluid behaviour in micro-engineered products containing a main channel, constriction and side channel bifurcations. Differences in fluid behaviour at the macro and micro-scales are discussed. The dynamic bulk fluid behaviour is characterized in terms of: (i) fluid properties, (ii) governing physics and (iii) microchannel geometric features. This paper presents such a modelling analysis fo T-microchannel fluid separator based on current assumptions that the shear rate of the flow is high, that is, higher than 100 s^{-1} for blood flow, and thus the fluid can be considered as a Newtonian flow; and the effect of any suspended particles is represented by a bulk viscosity and density. This provides the ability to investigate the bulk flow effects on the formation of a particle-depletion layer and design of the constriction and bifurcations. The overall aim is to predict the flow patterns, pressures and velocities throughout the micro-fluidic separator design and, in particular, to investigate the effect of the number of daughter channels and inlet–outlet conditions on the separation efficiency. Specific fluid dynamics and physical considerations are presented in the first part of the paper. In the second part, the simulation results are presented for two different flow activation patterns.

OBJECTIVES:

1. To study different micro-fluidic channels available in the literature.
2. To study the behaviour of fluidic flow in the channel to design the channel as per requirement.
3. To design different configuration of micro-fluidic channels.
4. To analyze and simulate micro-fluidic channel.
5. To develop physical model of micro-fluidic channels.

DESIGN CALCULATION:

- **Designing of Y-Mixer** :

Mix ethanol completely with water in a parallel micromixer with two inlets (Y-mixer) at room temperature. The flow rates of both ethanol and water are 10 µl/min. Determine the required length of the mixing channel, if the channel cross section has a dimension of 0.5mm×0.5mm .

Solution:

The diffusion coefficient of ethanol in water at room temp.(25°c) is 0.84×10^-5 sq.cm/s. The characteristic mixing length is the channel width w=0.5mm.The required mixing time is :

$$\tau = (w^2)/2D$$
$$= 297.61\text{sec.}$$

The average velocity of mixed liquid is :

$$u = (\dot{Q}_{water} + \dot{Q}_{ethanol})/A$$
$$= 1.33\times10^{-3}\ \text{m/s.}$$

Thus the required length of mixing channel is :

$$L = u \times \tau$$
$$=197 \text{ mm.}$$

- **Designing a Long Mixing Channel :**

The above mixing channel is to be designed with a meander shape to save lateral device surface. If the channel structure is to be placed inside a square area ,determine the dimension of area. Determine the no. of turns.

Solution:

We assumed that the channel was consume the same amount of area as the channel itself . The total surface area required for mixing channel is :

$$A = 2 \times w \times L$$
$$=197 \text{ sq.mm.}$$

The dimension of square area is :

$$a = \text{Sq.root of } A.$$
$$= 49.03\text{mm.}$$

Each turns consume 0.3mm × 0.5mm for walls and 0.3mm × 0.5mm for channel width .the total no. of turns is :

$$N = (a) /(4 \times 0.5)$$
$$= 9$$

Methodology:

1. Different micro-fluidic channels available in literature review will be studied such as T-shape, Y-shape, U-shape, S-shape, elliptical and spiral shape.

2. Behavior of fluidic flow in the channel will be studied as per requirement.
   a. Characterization of large surface to volume ratio of cross section and high aspect ratio of length to cross section of micro-channels will be studied. The effect of geometrical parameters on micro-channels with different shapes and dimensions of micro-channels will be studied by varying the channel configuration.
   b. The effect of surface roughness and change in the cross section of micro-fluidic channels on flow behavior will be studied. Also the effects of corners, sharp edges of micro-channels on fluid flow will be studied.

3. The different design parameters such as pressure, temperature, number of daughter channels, concavity etc. will be studied using analytical formulae available in the literature for different shapes. Different types of microfluidic channels will be designed as per requirements.

4. Numerical method such as finite element analysis, finite difference method and finite volume method will be studied for analysis and simulation of micro-fluidic channels. Some software packages will be used for analysis and simulation like CFD, Comsol-Multiphysics and MATLAB.

5. Physical model consists of preparation of negatives which the basic drawing which will be drawn on transparent paper with the help of drawing softwares such as Auto CAD 2011. Negative will be used to produce mask by applying photoresist from metal strip. U.V. light exposure is required to have negative drawing on the applied photoresist metal strip. Then we need to develop our metal strip to remove required photoresist from the metal strip. Then the metal strip can be kept in etching process. After etching mask gets ready.

Advantages:

1. The main advantage is that miniaturized components and processes use smaller volumes of fluid,thus leading to reduced reagent consumption.

2. It decreases costs and permits small quantities of precious samples to be stretched further (for example, divided up into a much larger number of screening assays)
3. Quantities of waste products are also reduced.
4. The low thermal mass and large surface to volume ratio of small components facilitates rapid heat transfer, enabling quick temperature changes and precise temperature control.

Disadvantage:

1. The devices which are required for microfludic channel fabrication are very expensive.