Numerical Modelling of Flow Boiling in Expanding Microchannel with Non-Uniform Heat Flux

Uday Kumar Alugoju¹, Rutuja Manmode¹, Satish K. Dubey¹, Arshad Javed¹

¹ Department of Mechanical Engineering, BITS Pilani, Hyderabad Campus, Telangana, India 500078.

Corresponding Author E-mail: p20150412@hyderabad.bits-pilani.ac.in

Abstract. Thermal management of high-power density electronic chips is vital for their safe operation and reliable performance. For efficient heat dissipation from electronic chip, liquid microchannel heat sink is preferred and is an effective option amongst the various available cooling technologies. In the recent past, flow boiling in microchannel has received considerable attention as an effective option for thermal management of high-power electronic devices. Flow boiling involves phase change and dissipates high heat fluxes for lower mass flow rate, when compared with single-phase flow. The commercial application of flow boiling microchannel for electronic cooling is hindered because of limited understanding of non-uniform wall temperature distribution, flow instabilities and flow reversal. In the recent past, more focus has been on numerical modelling and simulation of flow boiling in microchannel. In this work, numerical simulation of a 2D expanding microchannel heat sink with non-uniform heat flux has been considered and the simulations have been performed using ANSYS Fluent. Phase change heat transfer simulation has been performed using volume of fluid (VOF) model together with interfacial mass and energy transfer. The performance of expanding channel is then compared with the straight channel with similar heat flux distribution and the results are presented.

1. Introduction

Thermal management of electronic components is vital for their optimum performance, durability and reliability. Liquid cooled micro-channel heat sinks are an efficient and reliable method of thermal management in high heat generating electronic devices due to their high surface to volume ratio. As per the current trends, the heat flux generated by these devices is expected to rise even higher. Hence, flow boiling in microchannels can be used to dissipate even higher heat fluxes. However, the advantages are offset by instabilities encountered in case of micro-channels such as flow reversal, clogging of micro-channels due to explosive bubble growth and pressure oscillations [1]. There are few techniques available in the literature to mitigate these instabilities in microchannels like geometrical modifications, compressible volume/ vapor venting, varying flow property and surface modifications [2].

In geometrical modifications, Kandlikar et.al. worked on the improving flow stability using artificial nucleation sites [3]. Bogojevic et.al. experimentally studied the instabilities that occur during flow boiling in microchannels [4]. They analysed the various flow stability factors such as heat flux, mass...
flux and the channel geometry. Mukherjee and Kandlikar performed experimental studies on the effect of inlet constriction on flow boiling in microchannels [5]. Flow boiling in single diverging microchannel was experimentally investigated by Lee and Pan [6]. The performance of diverging channel was found to be better. They concluded that, the divergence angle is critical as, higher angles may result in flow reversal. Balasubramanian et.al experimentally compared the performance of straight and diverging channels [7]. Prajapati et.al. compared the performance of segmented channels with straight and diverging channel [8]. Based on their experimental results, they concluded that the performance of segmented channel was better than that of straight channel. Experimentally studying all the channel configurations is time consuming and difficult, hence numerical studies are being used. Zhuan and Wang have numerically studied the nucleate boiling in straight microchannels [9]. Prajapati et.al numerically investigated the sub-cooled flow boiling in segmented channels using ANSYS Fluent and VOF (volume of fluid) method [10]. They studied the heat transfer and flow characteristics in the segmented channel. Abedini et.al, have numerically investigated the vapour fraction in flow boiling microchannels with Nano fluids [11]. They found the correlation between axial volume fraction to the Nano particle concentration. Liu et al. studied the bubble interaction and meager during flow boiling in microchannels [12]. Bahreini et.al numerically simulated flow boiling in vertical minichannel under micro gravity conditions [13]. However, in most of the practical situations, the applied heat flux is not uniform, i.e. a non-uniform heat flux condition would be prevalent. In the recent study, Lorenzini and Joshi presented a 3-D numerical simulation of flow boiling in straight microchannels with non-uniform heat flux condition [14]. They have coupled VOF model with a phase change model to account for interfacial mass and energy transfer and studied the bubble growth pattern.

The objective of this study is to do a comparative study between straight and diverging channel in non-uniform heat flux situation which resembles the practical scenario. This paper also analyses the performance of divergent microchannel in non-uniform heat flux situation wherein variation in onset nucleation time, coolant temperature, pressure fluctuations and bubble dynamics in the channel were studied.

2. Numerical Simulation

2.1. Computational Domain

Two-dimensional diverging channel and straight channel with non-uniform heat flux condition are considered for the numerical simulation. The straight channel has a width of 0.4 mm, while diverging channel has an inlet width of 0.3 mm and an outlet width of 0.522 mm. Both the channels have length of 25.7 mm. The detailed geometry is shown in figure 1.

2.2. Governing Equations

The two-phase simulations for flow boiling microchannels were performed in the computational domain. Water is used as coolant with liquid as primary phase and water vapour as secondary phase. The governing equations: continuity, momentum and energy are as follows [10].

*Continuity equation:*
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \]  

**Momentum equation:**

\[ \rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \nabla \left[ \mu (\nabla \mathbf{v} + \nabla \mathbf{v}^T) \right] + S \]  

where ‘\( S \)’ is the surface tension force and is calculated by using continuous surface force (CSF) model.

\[ S = -\sigma \kappa \nabla \alpha \]  

The bubble interface curvature (\( \kappa \)) is given as follows:

\[ \kappa = \frac{\nabla \alpha}{|\nabla \alpha|} \]  

**Energy equation:**

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

Enthalpy \( E \) is calculated as follows:

\[ E = \sum_{i=1}^{i=2} \alpha_i \rho_i E_i \]  

Where \( p \) is pressure, \( \mathbf{v} \) is velocity, \( \alpha \) is density, \( \alpha \) is volume fraction, \( S \) and \( S_h \) are the source terms used to model the surface tension force and the change in enthalpy in the momentum and energy equations respectively. \( k_{eff} \) is the effective thermal conductivity. The density and viscosity are given as follows:

\[ \rho = \rho_1 + \alpha (\rho_2 - \rho_1) \]  

\[ \mu = \mu_1 + \alpha (\mu_2 - \mu_1) \]  

Where \( \alpha \) is density, \( \mu \) is viscosity and subscript 1 and 2 represent the liquid and vapour phase respectively.

Volume of fluid (VOF) model is used to track the interaction between the primary and the secondary phase. Where the distribution of void fraction is given as below:

\[ \frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{v} \nabla \alpha) = \frac{S_h}{\rho_1} \]  

The source term \( S_h \) takes care of the changes in energy due to phase transition, and is given as

\[ S_h = \dot{m} h_{fg} \]  

\( \dot{m} \), is the rate of mass transfer during phase change and it depends on the saturation temperature of the liquid which is given as follows

\[ \dot{m} = c (1 - \alpha) \rho_1 \frac{(T - T_{sat})}{T_{sat}} \quad (T \geq T_{sat}) \]  

\[ \dot{m} = c \alpha \rho_2 \frac{(T_{sat} - T)}{T_{sat}} \quad (T_{sat} \geq T) \]  

\( T_{sat} \) is the saturation temperature of the liquid and \( c \) is the relaxation parameter responsible for correct trend of mass transfer.
2.3. Boundary conditions
Both straight and diverging channels are simulated for mass flow at the inlet and pressure outlet boundary conditions. The applied mass flux at inlet is 200 kg/m²s. Copper is considered as solid domain. Water is considered as the coolant and it enters the channel in liquid phase at 300 K. Therefore, volume fraction of vapour phase at inlet is considered as zero. The channel has been divided into 24 segments and non-uniform heat flux has been applied based on the power map of dual core electronic architecture which is shown in table-1 [14] [15]. Numerical simulations have been performed on ANSYS Fluent. Volume of Fluid (VOF) along with implicit scheme was used to simulate flow boiling phenomenon. Pressure Implicit with Splitting of Operators (PISO) has been implemented for coupling pressure and velocity field. The time steps of $10^{-5}$ seconds has been used to maintain numerical stability.

| SNo | $q''$ (W/cm²) |
|-----|---------------|
| 1   | 0             |
| 2   | 25            |
| 3   | 115           |
| 4   | 125           |
| 5   | 125           |
| 6   | 115           |
| 7   | 35            |
| 8   | 35            |
| 9   | 67.5          |
| 10  | 67.5          |
| 11  | 72.5          |
| 12  | 85            |

2.4. Validation
In order to validate the numerical implementation of flow boiling, simulation was performed for 2D segmented channel with uniform heat flux similar to the conditions given by Prajapati et.al. [10].
In this validation study a 2D segregated channel with a uniform heat flux condition of 315 kW/m² is simulated for a mass flux of 195 kg/m²s. The numerical results are obtained from the simulations for a duration of 0.25 sec and compared with the published experimental results. Figure 2 shows the comparison of local convective heat transfer coefficient along the length of the channel. It can be observed that the present numerical simulation results are in agreement with the published results.

3. Results and discussion
A two dimensional straight and a diverging channel with a non-uniform heat flux has been simulated for a mass flux of 200 kg/m²s. Non-uniform heat flux condition has been applied to both the cases. The performance of both the channels has been compared and presented here.

**Bubble growth in the straight microchannel**
The onset nucleate boiling starts at a point where the wall temperature is higher than the liquid saturation temperature. Figure 3 shows the bubble growth in straight microchannel at three different timesteps t=2.4, 5.4 and 7 milliseconds. Here the vapour phase of the coolant is indicated by red colour and the liquid phase of the coolant is indicated by blue colour. Form the figure, it can be observed that the phase change of the coolant starts at t=2.4 millisecond and formation of bubbles can be seen at t=5.4 milliseconds. At t=7 milliseconds the generated bubbles can be seen detaching from the wall and flowing into the stream of the coolant. This detachment happens when, the buoyancy forces are stronger than the surface tension. Further, this leads to bubble-bubble interaction and bubble coalescence. Here, it can be observed that the bubble generation pattern is dependent on the heat flux distribution.

**Figure 3**: Bubble growth in straight channel at (a) 2.4, (b) 5.4 and (c) 7 milliseconds

**Effect of divergence on bubble pattern**
Figure 4(a), shows the bubble generation in the diverging micro channel segment from x=16.7 mm to 25.7 mm for t=20 milliseconds. Figure 4(b) shows bubble pattern in straight micro channel segment from x=16.7 mm to 25.7 mm at t=20 millisecond. For the similar timestep it can be observed that the number of bubbles generated is more in the diverging channels when compared with the straight channel. But more bubbles of larger diameter can be observed in the straight channel. This is due to the higher convection rate in the straight channels, less amount of heat is present to generate new bubble. Hence, less bubbles are generated with larger diameter in the case of straight channel. Also, in the channel with lower hydraulic diameter, the nucleation time is decreased. It is observed that the smaller diameter bubbles are detached from the wall as they move downstream and interact with other bubbles and form a coalesced bubble with a larger diameter.
Figure 4(a): Volume fraction in diverging channel at x=16.7mm to 25.7mm (t=20 millisecond).

Figure 4(b): Volume fraction in Straight channel at x=16.7mm to 25.7mm (t=20 millisecond).

Effect of divergence on coolant temperature and pressure in the channel

One of the main issues of the flow boiling in microchannels heat sink is temperature and pressure fluctuations. Within a span of a few milliseconds, there can be significant variation in the pressure and temperature of the microchannel. These fluctuations will lead to thermal load, localised heating in the channel and structural loads on the channel walls. So, an attempt was made to understand the temperature and pressure fluctuations in both straight and diverging microchannel heat sink at the same time. Figure 5(a) shows the variation in the coolant temperature along the length of the central axis in diverging channel at t = 20 milliseconds. It can be observed that at the inlet of the channels the coolant temperature is at 300 K. As we move downstream of the channel, depending on the heat flux condition, the coolant temperature varies accordingly. The maximum temperature along the central length of the channel is 350 K. Hence it can be observed that, the generated bubbles are mostly at the channel walls and are yet to move towards the channel centre.

Figure 5(a): Variation in the coolant temperature along the center of the length in diverging channel at t=20 milliseconds
Figure 5(b) shows the variation in the coolant temperature along the central axis in the straight channel at t= 20 milliseconds. Here, the coolant enters the channel at 300 K and as we move downstream along channel there is a variation in the channel temperature. At the exit of the channel, the fluid temperature is around 380 K indicating the presence of bubble along the central axis in the straight channel at t= 20 milliseconds. Figure 5 (c) shows the variation of the coolant temperature in straight channel at t=30 milliseconds. Few bubbles can be observed at the central axis of the channels but most of the bubbles are generated along the channel wall and very few are present at the channel centre. This is due to the lower time frame considered in this study.

**Figure 5(b):** Variation in the coolant temperature along the center of the length in straight channel at t=20 milliseconds

**Figure 5(c):** Variation in the coolant temperature along the centre of the length in straight channel at t=30 milliseconds
As discussed previously, the pressure fluctuation plays a vital role in the flow and heat transfer characteristics in the channel. An attempt was made to understand the pressure distribution in both straight and diverging channels. The pressure contours of straight microchannel segment from x=20.7 mm to 25.7mm at t=20 millisecond is shown in figure 6(a). It can be observed that the pressure in the channel is significantly influenced by the presence of bubbles. The area surrounding the bubble has a lower pressure when compared to the pressure inside the bubble. Figure 6(b) shows the pressure contours of diverging microchannel segment from x=20.7 mm to 25.7 mm at t= 20 milliseconds. Pressure distribution trend in the diverging channel is found to be similar to straight channel. However, pressure magnitudes in diverging channels differs significantly from that in straight channel, as bubble generation patterns are different in the latter. Hence, it can be observed that on varying the hydraulic diameter of the channel the bubble pattern and pressure of the channels is also varies. Here, further studies have to be conducted to understand whether the observed phenomenon is due to diverging channel or due to applying non-uniform condition. This can be done by analysing the effect of divergence angle and non-uniform heat flux condition separately.

4. Conclusion
Two-dimensional numerical simulation of flow boiling in straight and diverging microchannel for a non–uniform heat flux condition has been studied. Bubble pattern and effect of divergence on the performance of the microchannel has been studied. It was observed that, more bubbles with smaller diameter are observed in the diverging channels while less bubbles with larger diameter are observed in the straight channel. Temperature and pressure fluctuation are observed in both the channels. The nucleation time was also found to be different in both the channels. Lesser nucleation time was observed in straight channels. Further studies have to be carried out to understand the individual impact of non-uniform heat flux distribution and the impact of divergence angle on the performance of flow boiling in microchannel heat sink. 3D simulations also have to be performed, to understand the conjugate effect which also depends on the heat flux distribution.
References

[1] Kandlikar S G Willistein D A & Borrelli J 2005 Experimental Evaluation of Pressure Drop Elements and Fabricated Nucleation Sites for Stabilizing Flow Boiling in Minichannels and Microchannels In ASME 3rd International Conference on Microchannels and Minichannels Part B cont’d pp 115–124 ASME

[2] Prajapati Y K, Bhandari P. Flow boiling instabilities in microchannels and their promising solutions--A review. Experimental Thermal and Fluid Science 2017 Nov 1;88:576-93.

[3] Kandlikar S G Kuan W K Willistein D A & Borrelli J 2006 Stabilization of Flow Boiling in Microchannels Using Pressure Drop Elements and Fabricated Nucleation Sites Journal of Heat Transfer 128 4 389

[4] Bogojevic D Sefiane K Walton A J Lin H and Cummins G 2009 Two-phase flow instabilities in a silicon microchannels heat sink International Journal of Heat and Fluid Flow 30-5 854–867

[5] Mukherjee A & Kandlikar S G 2009 The effect of inlet constriction on bubble growth during flow boiling in microchannels International Journal of Heat and Mass Transfer 52-21 5204–5212

[6] Lee P C & Pan C 2008 Boiling heat transfer and two-phase flow of water in a single shallow microchannel with a uniform or diverging cross section Journal of Micromechanics and Microengineering 18 2 25005

[7] Balasubramanian K Lee P S Jin L W Chou S K Teo C J and Gao S 2011 Experimental investigations of flow boiling heat transfer and pressure drop in straight and expanding microchannels A comparative study International Journal of Thermal Sciences 50-12 2413–2421

[8] Prajapati Y K Pathak M and Kaleem Khan M 2015 A comparative study of flow boiling heat transfer in three different configurations of microchannels International Journal of Heat and Mass Transfer 85 711–722

[9] Zhuan R and Wang W 2010 Simulation on nucleate boiling in micro-channel International journal of heat and mass transfer 53 (1-3) 502-512

[10] Prajapati Y K Pathak M and Khan M K 2017 Numerical investigation of subcooled flow boiling in segmented finned microchannels International Communications in Heat and Mass Transfer 86 215-221

[11] Abedini E, Zarei T, Rajabnia H, Kalbasi R, Afrand M. Numerical investigation of vapor volume fraction in subcooled flow boiling of a nanofluid. Journal of Molecular Liquids 2017 Jul 1;238:281-9.

[12] Liu Q, Wang W, Palm B. Numerical study of the interactions and merge of multiple bubbles during convective boiling in micro channels. International Communications in Heat and Mass Transfer 2017 Jan 1;80:10-7.

[13] Bahreini M, Ramiar A, Ranjbar AA. Numerical simulation of subcooled flow boiling under conjugate heat transfer and microgravity condition in a vertical mini channel. Applied Thermal Engineering 2017 Feb 25;113:170-85.

[14] Lorenzini D and Joshi Y K 2018 Computational Fluid Dynamics Modeling of Flow Boiling in Microchannels with Nonuniform Heat Flux Journal of Heat Transfer 140-1 011501

[15] Koo J M Im S Jiang L and Goodson K E 2005 Integrated microchannel cooling for three-dimensional electronic circuit architectures Journal of heat transfer 127-1 49-58