Computational analysis for different characteristics of the circular microchannel under low Reynolds number fully developed flow conditions

Karan Veer Singh, Rohit Soni, Animesh Singhai

Department of Mechanical Engineering
(Trinity Institute of Technology and Research, RGPV Bhopal)
(Kokta Bypass Rd, Near Hindustan Petrol Pump, Patel Nagar, Bhopal, Madhya Pradesh 462021)

---***---

Abstract - The present research works aims examine the flow and thermal non-uniformity in a circular micro channel (CMC). A concept of variable size approach is employed to mitigate flow and thermal non-uniformity in rectangular CMC. The heat dissipation requirement of an electronic chip of processor is deliberated in the present study. It is found that proposed design improves flow rate and heat dissipation with increase in diameter. It is found that increasing in diameter, the heat dissipation rate increase.

Keywords – Micro-channel; electronic devices; flow rate; chip cooling

INTRODUCTION

The rapid growth of high-density power electronic, with the increased miniaturization of microelectronic devices and processing speed, thermal issues are more and more affecting overall electronic packaging and system capabilities. Problems related to heat dissipation of microelectronic devices have brought need of supplementary research and development. Microelectronic device performance and reliability are known to increase when effective temperatures are kept below 40°C [1]. Keeping the temperature conditions in focus, during last two decades research on cooling of electronic devices has increased multi-fold. Earlier, heat removal from electronic devices was done by using forced air convection. With the increase in power and speed of electronic devices, the amount of heat to be removed has increased. This requires enhancement in heat transfer coefficient [2]. Owing to upper limits of achievable heat transfer coefficient, the other modes, namely pool boiling and flow boiling using dielectric fluids began to receive attention for microelectronic device cooling. The methodology can be adopted in system integration efficiently.

D. B. Tuckerman et al (1981) has been investigated, convection heat-transfer coefficient 'h' between substrate and coolant is considered to be the primary impediment to achieving low thermal resistance. Cooling viscosity determines the use of high-aspect-ratio channels to maximize the minimum practical channel width and surface area, which inspires the thermal support based on these ideas, new very compact water-for silicon- Cooled Integral Heat Ill Circuit has been designed and tested.

Gregory Swift et al (1985) has seen that the construction of a highly compact cross-flow heat exchanger is done by kiln-grazing, simultaneously stacking hundreds of stainless-steel sheets and water flowing through a heat exchanger Measurement of heat transfer between liquid propylene. Exchangers are in excellent agreement with calculations based on geometry and fluid properties.

Ian Papautsky et al (2001) summarized in the field of experimental research endeavor on the microscale single-phase internal fluid flow and the issue related to the examination of the flow of microscale has been discussed. Micro-channel surface roughness appears to increase, it seems to intensify with higher surface roughness. Laminar friction appears to be 20% higher than theoretical predictions for supported water flow unrest transition, it has been detailed that the ray is not reliable. It was reported that the surface of the micro-channel appears to increase roughness which is sharper with high surface roughness. Laminar friction appears to be nearly 20% more consistent than the theoretical prediction for the flow of water. No reliable range of rays has been detailed for unrest transition. Some data showed that the disturbance is as low as the Micro Scale [Re =500]. At present, precise prediction of droplets of pressure in micro-channels is not possible for non-drop flows, sometimes pressure drops can be estimated using conventional theory and correlations.

Dongliu et al (2002) experienced that the fluid flow in the micro-channel have been both observational and scientific. The micro-channels with hydraulic diameters from 230 to 6500 on the Reynolds number from 244 to 974 dimensions were used. In these micro-channels, the pressure is measured within the drop-seat. The observational estimation of the pressure drop is in comparison to the scientific prediction. The result of the last but least mathematical simulation has not fulfilled the agreement with empirical measurements. The advancement of totally turbulent flow is turned into micro-channels, in the shape of less subtle channels that can have a critical effect on the improvement of unrest, which is recommended by the scale of Kolomorov length. In micro-channels, due to flow visibility, due to contamination, the beginning of unrest was qualitative. A great common assertion between the result of numerical simulation and experimental measurements.

Wai Keat Kuan et al (2006) inspected, the experimental test section has six parallel rectangular micro-channels, each of which has a cross-sectional area of 1054 × 197 microns. The fluctuation effect on the boiling heat transfer flow in micro-channels has been seen as the working fluid of water. Flow restrictions were introduced on the inlet of each micro-channel to prevent flow restriction process and to avoid backflow phenomenon. Results are shown in comparison to unrestricted flow arrangement. The effect of flow inhibitors on heat transfer performance during boiling the flow in micro-channels was studied. Flow blockers act as a physical pressure drop element (PDE) that is being studied. The result from the case without PDE is compared to 6.1% of PDE. The heat transfer result is for the large flow of 362.9 kg / m2 and for 144.4 kg / m2 and 362.9 kg / m2. Heat transfer performance increases to 144.4 kg / m2 by using a header with 6.1 PDEs. Using headers with 6.1% PDE has reduced the heat transfer performance. In the case of 6.1%, PDE does not improve heat transfer performance for low mass flow rates. The result of heat transfers for a large flow of 362.9 kg / m2 and 144.4 kg / m2. In the case of 362.9 kg / m2 using the headers, heat transfer performance has improved 6.1% PDE and 6.1% PDE did not increase the heat transfer performance for low mass flow rates.

H. Y. Zhang et al (2009) have been seen that some parameters affect the microchannel, perhaps it is a new phenomenon of micro-channels, namely as the vortex start and transition to turbulent flow for the rain old number, is less compared to the flow through conventionally shaped tubes. , Other aspects come from familiar events, which are generally neglected for flow and heat transfer through the tube with the flow, such as flow slip, Sticky wastage, and compressed flow.

Liang Gong et al (2011) performed parametric study of three-dimensional laminar fluid flow and heat transfer attributes in micro-sized wavy channels by varied the heavy feature amplitude, wavelength, and aspect ratio for different Reynolds Numbers between 50 and 150. It was found that a heat flux of 47w/cm2 accepted for a chip size of 1 cm2 found at the channel center with respect to flow direction. Both Nu and ΔP increase with A and Re.

Yanhui Han et al (2012) studied that Micro-Channel Heat Exchanger (MCHX) has applied in heating, ventilation, air conditioning, and refrigeration because it is a more effective heat transfer rate and a more compact measure and lower cost. In this study, the qualities of micro-channel heat transfers and fluid dynamics have been summarized and the strategies of adjustment namely Geometry and thermodynamic execution and the preferences and impediments of MCHX were examined. It was found that the micro-channel heat exchanger has been executed more broadly in the refrigeration and air conditioning industry. Before designing the micro-channel, pressure loss and heat transfer characteristics should be accurately estimated. In this manner, it is accepted that in the analysis of micro-channel heat exchanger functioning in-depth, optimize the heat transfer size and solve existing problems in fabricating and application.

Amirah M. Sahar et al (2014) saw single-phase flow and conjugate heat transfer in a rectangular microchannel, which is aligned with a glass plate in the copper block of the heater, in which numerical simulation fluent 14.5 is used. The water used in this study fluid has been used and the hydraulic diameter of the rectangular channel is 0.561 mm and the length is 62 mm. Mathematical results have been validated using experimental data and existing traditional theory. The result showed that there was an important difference between the 3D thin wall and the fully conjugated model and it demonstrated that the numerical plan was employed, which was capable of providing a precise simulation of flow and heat transfer even in the microchannel. It was analyzed that four models specifically 2D, 3D thin wall (one side hot), 3D thin wall (three-way hot) and 3D fully conjugated heat transfer models. Experimental and mathematical results show that the transition to laminar turbulence occurs again in 1600 – 2000. 3D thin-wall model predicts exploratory value with the sensor of the 3D conjugate model. The divergence between empirical values and conjugate models may be due to the fact that the conjugate effect is not taken into consideration in the experimental data reduction process. It was found that exploratory and numerical results showed that Re-1600-2000 transmitters of laminar in the turbulent. Experimental value with excellent agreement compared to the 3D conjugate model in 3-D thin-wall model prediction. The period between the conjugate model and exploratory values may be due to the fact that the conjugate effect is not taken into consideration in the experimental and data reduction process.

Ravindra Kumar et al (2014) observed that in spite of the fact that an exploratory investigation has been done on the validity of classical correlations based on traditional size channels for the prediction of flow and heat transfer behavior of semi-circular micro-channels in single-phase fluid flow. The experiments have been done with the disintegrating water cooling and Reynolds number 238 to 1250. Experimental results have shown that the results of the conventional theory were applicable to the flow of fully developed laminas for the range of our experiments. Classical correlations were evaluated for micro-channel, with the perception of continuous wall heat flux, the average Nassel number obtained for micro-channel. It was investigated to make an accurate estimation of heat transfer rates and consider the effect of admission effect, conjugate effect, axial heat conduction, and boundary conditions. Empirical investigations have been done for the friction factor in the single-stage liquid flow inside the semi-circular micro-channels. The results of the experiment show the agreement with traditional hydraulic theory and see that the flow of fluid inside the microchannel fully develops the flow of laminar in the range of experiments (238 <Re <1250). It was found that experimental results have found that a good deal has been reached with traditional hydraulic theory and it has been observed that the flow inside the micro-channels is the flow of the fully developed experiments (238 <Re <1250) it shows. The experimental value of the Nusselt number in the lower Reynolds number is slightly different from the correlations developed by SHAH & LONDON and Sieder-Tate. It is observed near the estimated result by correlation developed by Sieder-Tate and confirms that the flow is evolving together. The prediction of the result by numerical simulation for MCHS with a very thin substrate agrees well with experimental results and suggests that further analysis is needed to develop suitable new correlations for semi-circular cross-sectional MCHS. It is suggested that low Reynolds number flow is better.

T.G. Karayiannis et al (2016) stated that due to the fast increment in the performance and commute of electronics and high-power equipment, heavy heat flow values need to be successfully broken. The author has seen, the potential application of boiling flow in micro-channels to highlight the challenges in thermal management for each application. The author presented exploratory investigate on the following fundamental factors - the flow of flow in rectangular multi-channels and single tubes to examine the definition of micro-channel. Flow pattern and heat transfer mechanism fluctuation and their impact on reversal and heat transfer rates, impact of channel surface characteristics, noteworthy heat flow prediction. The correlation for the prediction of the flow pattern transition and the heat transfer coefficient in small micro-diameter tubes was developed recently. Flow boiling in a microchannel heat sink is very promising for many applications such as the computer and IT industry high power semiconductor equipment. The average heat flow is predicted to reach 2-4.5 MW / m2 in a computer chip, which contains local hot spots in the range of 12-45 MW / m2. In high power semiconductor devices, the heat flow can be 6.3 to 50 MW / m2 at the chip level. The instability of the flow and reversal is characterized by researchers rapidly developing bubbles. The authors suggest that channel surface characteristics, surface wettability, conjugated heat effects, and inlet sizes and outlet manifolds are important parameters that affect reversal and flow instability. Various heat and channel flux of channel surface characteristics have a significant impact on behavior on local heat transfer coefficients. Some studies analyze the effects of surface fluctuation on boiling the flow in the micro-channels and examine the contradictory effects of the impending tilt. Correlation designs were created to predict heat transfer and flow patterns in small micro-diameter channels. There is an agreement among analysts on the Flow Critical Heat Flux on mass flow effect, fluid properties, length ratio, and inlet subcooling. It was found that there is no common assertion on the criteria utilized for the definition of macro-mini /mini/micro-channels. Within the two-stage flow, all physics-based criteria were given the assignment of fluid properties and flow parameters, which depend on the conditions of the real parameters. Some studies analyzed the effect of melting the surface on boiling the flow in microchannels and saw the contraction effect on the provocative urine. A common assertion between researchers on the effect of mass flux and fluid properties, critical heat flux over the flow of the diameter and inlet subcooling. Designs were developed to predict flow patterns and heat transfer in small micro-diameter channels.

Hasan Nihal Zaidi et al (2017) stated that in the presence of a transverse magnetic field and internal heating generation to the authors, the issue of fully developed free convection flow of electrostatic fluid in a tilted micro-channel has been inspected and an explanatory arrangement has been found for velocity and temperature profile. The authors have considered the temperature jump and velocity slip on the microchannel wall and discussed the impact of diverse parameters with velocity and temperature profile along with the skin friction parameter and graphs on the

Nusselt number. It was found that when the Knudsen number (Kn) increases the velocity of the particle growth, then the coefficient of heat generation and the interaction parameter (in) decreases with the increase of the Hartman number.

M.R. Ozdemir et al (2018) studied that a comprehensive review of single and two-phase pressure drop characteristics and flow-boiling instability in micro-channels to exclude line discrepancies in the literature. Large-scale flow effect, heat flow, exploratory conditions Channel geometric parameters (hydraulic diameter, aspect ratio) have been studied critically, including past and recent researchers. Two-phase pressure drop in the micro-scale has found similar properties with traditional-scale channels and the flow boiling instability has affected the performance of the system and in this study, the unstable flow for the reliable design of the microscopic heat boiling conditions have been identified Exchangers. It was said that for single-phase pressure drop. There is a need to consider the loss of inlet and exhaust pressure, the single-phase friction factor strongly depends on the characteristics of the surface of the channel, due to the transition size effect between the laminar turbulence in micro-channels can be the traditional channel.

S.M. Chan et al (2018) imitated the impact of 10-micron transitional micro-channel on the thermo-hydraulic display for single-phase flow in semi-circular cross-sectional surfaces, in which the position of the boundary for the temperature of the wall is consistent 350 k. It has been shown that with the Reynolds number, deans buy vortex and increase the heat transfer in the zone of spiral turn. The study proposed that a hydraulic diameter channel of 10 microns can improve heat transfer performance. Appropriate for 10-diameter hydraulic diameter channel for single-phase flow in spiral transitional micro-channel with a semi-circular cross-section.

Tao Wang et al (2018) described that in the single-phase microchannel heat sink (MCHS) temperature fluctuation, which uses the integrated temperature sensor with deionized water in the form of coolant. After the investigation, the result showed that the temperature fluctuation in temperature is not negligible. The motivation for fluctuations in temperature is based on both simulation and experimentation. It has been found that the effect of fluctuations in the inlet temperature is global, the temperature changes at different places at once and the impact of the gas bubble is localized, where the temperature changes are not synchronized at distinctive locations. The temperature gradient of the microchannel is linearly dependent on temperature fluctuations, where large temperature gradient increases fluctuations in high temperatures of MCHS.

Problem Definition of the Present Work

The micro-channel cooling technique appears to be a viable solution to high heat dissipation requirements of the micro-electro-mechanical system (MEMS) devices. The thermal design of these devices is a key issue that needs to keep the operating temperature at below the acceptable limit. Hence, the aim of the present work is to improve the thermal performance and reliability of these devices, by maintaining the operating temperature below the critical temperature say 85°C. The numerical study was conducted on single-phase forced convection in micro-channel using distilled water (DW) as a coolant. The set of numerical simulation was performed to measure the surface temperature of micro-channel at fixed volume flow rates in the range of 10–25 ml/min. It is also design to improve the uniformity of temperature distribution on the surface of MEMS devices and improves the cooling efficacy of micro-channel using as DW as a coolant.

Geometry of numerical model

A circular micro channel's computational domain is showed in 3-dimensional (3D) in figure. The geometry exists of a pipe. A centre line and inlet and outlet boundaries, the diameter d and length l of the pipe are specified in figure

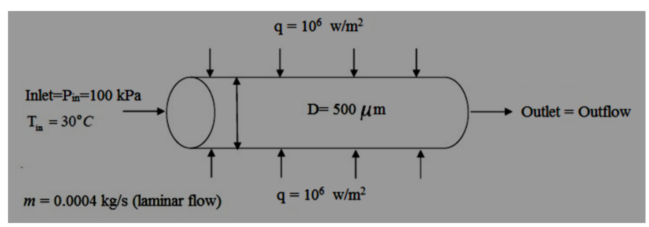

Figure 1 Schematic Diagram of circular micro-channel

Mathematical modeling equations

Equation of continuity, momentum equation, and energy equation (ANSYS Fluent 15.0) are included as single-phase model equations. The continuity and momentum equations calculate the velocity vector. The energy equation calculates temperature distribution and the wall heat transfer coefficient. The equation for conservation of mass, or continuity equation, can be depicted as:

Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho_\omega}{\partial t} + \nabla.(\rho_\omega \vec{v}) = Z_m \qquad (1)$$

Equation (1) is the general form of the mass conservation equation, and is valid for both Incompressible and compressible flows. The source Z_m is the mass added to the continuous Phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

Momentum Conservation Equation

Conservation of momentum in an inertial (non-accelerating) reference frame is explained by:

$$\frac{\partial}{\partial t}(\rho_w \vec{v}) + \nabla.(\rho_w \vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho_w \vec{g} + \vec{F} \qquad (2)$$

Computational Fluid Dynamics Model Equations

Where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor (described below), $\rho_\omega \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively.

$\vec{F}$ also contains other model dependent source terms such as porous-media and user-defined sources.

The stress $\bar{\bar{\tau}}$ is given by:

$$\bar{\bar{\tau}} = \mu\,[\,(\nabla\vec{v} + \nabla v^{-t}) - \frac{2}{3}\nabla\vec{v}\, I\,] \qquad (3)$$

Where μ is the molecular viscosity, I is the unit tensor, and the second term on the right Hand side is the effect of volume dilation.

Energy equation

Energy equation is solved by ANSYS FLUENT in the following form:

$$\frac{\partial}{\partial t}(\rho_w E) + \nabla\left(\vec{v}\,(\rho_w E + p)\right) = \nabla\,(K_{eff} - \nabla \mathrm{T} - \textstyle\sum_j h_j \vec{J_j} + \left(\overline{\overline{\tau_{eff}}}.\overrightarrow{v)}\right) + s_h \qquad (4)$$

Where K_{eff} is the effective conductivity (K+K_t) where K_t is the turbulent thermal conductivity, defined according to the turbulence model being used), and J_j is the diffusion flux of species J. The first three terms on the right-hand side of equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. s_h includes the heat of chemical reaction, and any other volumetric heat sources.

In Eq. (4),

$$\mathrm{E} = \mathrm{h} - \frac{p}{\rho} + \frac{v^2}{2} \qquad (5)$$

Where sensible enthalpy h is defined for ideal gases as

$$h = \textstyle\sum_j Y_j h_j \qquad (6)$$

Y_j is the mass fraction of species j.

$$h_j = \int_{T_{ref}}^{T} C_{p.j}\ \mathrm{d\,T} \qquad (7)$$

T_{ref} is used as 298.15 K.

Boundary Conditions

Boundary condition applied in these numerical simulations are given below:

- A no slip boundary condition.
- Inlet pressure = 100kpa,
- inlet temperature =300c,
- mass flow rate =0.0004kg/s (laminar)
- Heat flux =106 w/m2. Outlet =outflow.

Results and discussion

Meshing of geometry: structured meshing method has done in Ansys workbench was used for meshing the geometry 70125 nodes were generated. The 3D geometry of circular micro channel with structured mesh shown in figure.

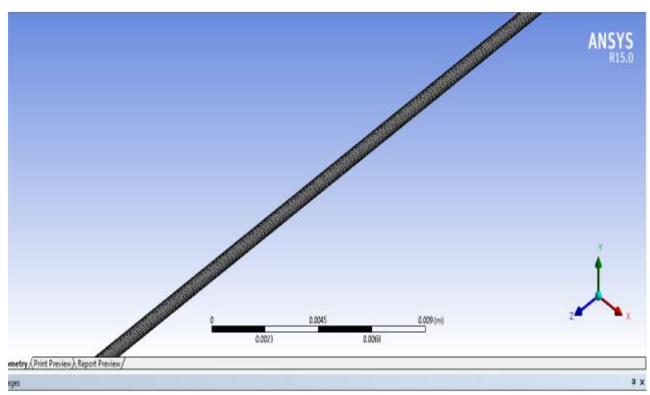

Figure 2. Meshing of Geometry of circular micro-channel

Variation of the temperature along the axial length

Figure 3 shows that the variation in water temperature at center line in laminar regime along with axial length of micro channel is gradually increasing from 303 k up to 333k.

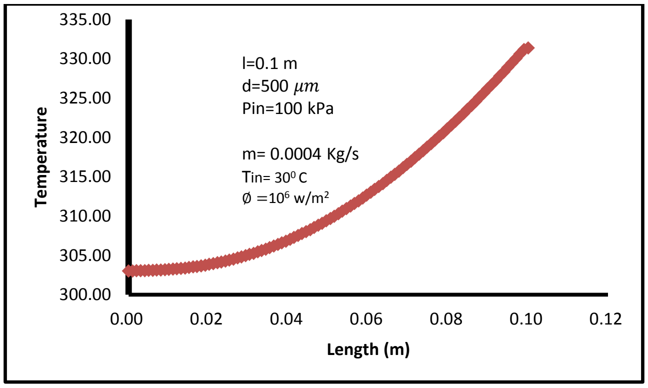

Figure 3. Temperature profile at center line in micro channel

Variation of the enthalpy along the axial length

Figure 4 Shows that enthalpy of water is gradually increasing along the axial length of micro channel .it is increasing from the point 20000 j kg^{-1} up to 140000. The enthalpy of water is showing of center line at laminar state with the axial length of micro channel.

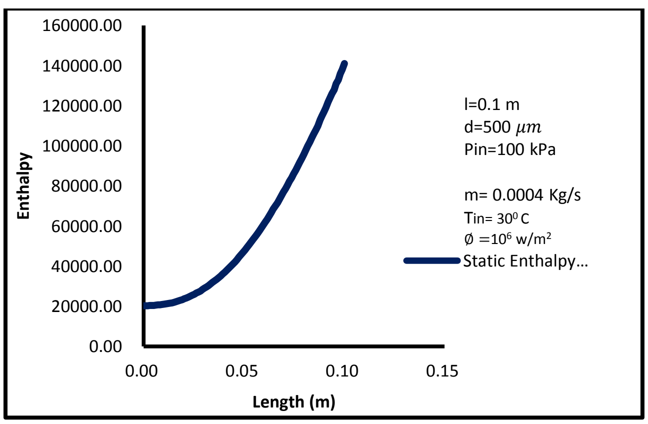

Figure 4 Enthalpy profile at center line in circular micro channel

Variation of the velocity along the axial length

Figure 5 shows that velocity of water in laminar regime at center line in micro channel is increasing from 2.00 ms^{-1} up to 3.50 ms^{-1} then it becomes constant so it is showing initially get the fully developed region.

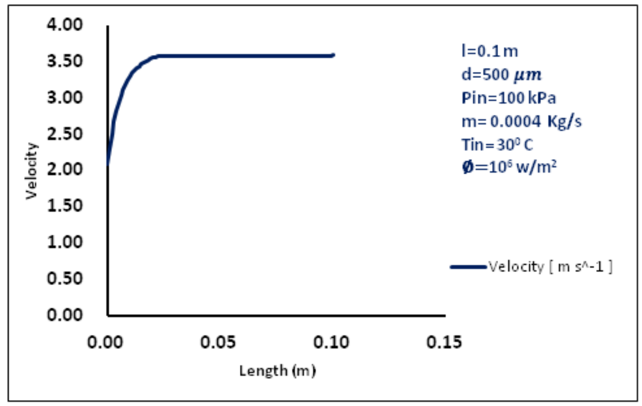

Figure 5. Velocity profile at center line in circular micro channel

Variation of the Reynolds number along the axial length

Figure 6 Shows that the Reynolds number of water is gradually increasing at laminar regime of axial length of micro channel's center line. It increases from 140 up to 250 then it becomes constant along with the axial length of micro channel.

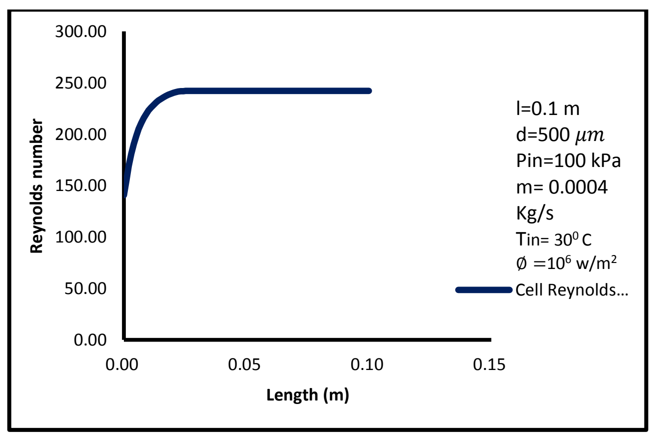

Figure 6. Reynolds number profile at center line in circular micro channel

Effect of different diameter on the enthalpy in circular micro channel

Figure 7 shows that the variation in enthalpy of water at center line in circular channel, with 500 micro diameter shows is increasing from 20000 up to 150000. It shows the 400 micro diameter is increasing from 20000 up to 120000 and this shows the 600 micro diameter is increasing from 20000 up to 100000. So, we find that the highest enthalpy 500 micro diameter then 400 micro diameters then 600 micro diameters of circular micro channel. It can be clearly seen that the enthalpy of the micro-channel has been increased with in increasing in the diameter.

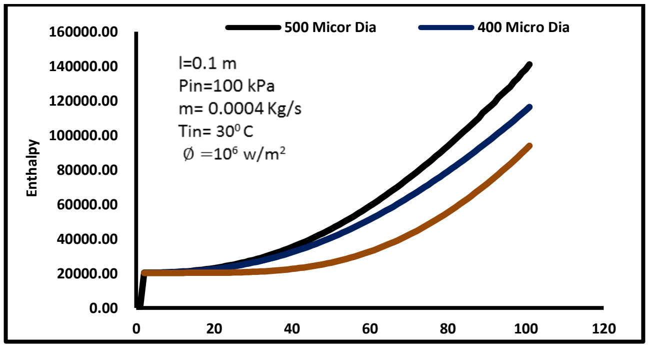

Figure 7 Effect of diameter on the enthalpy profile at center line in circular micro channel

Effect of different diameter in the velocity in circular micro channel

Figure 8 that velocity of water in laminar regime at center line in micro channel higher increasing in that 400 micro diameter 6 ms-1 then 500 micro diameter 3.6 ms-1 then 600 micro diameter 2.62 ms-1 then it becomes constant so it is showing initially get the fully developed region. It can be clearly seen that the velocity of the micro-channel has been increased with in increasing in the diameter.

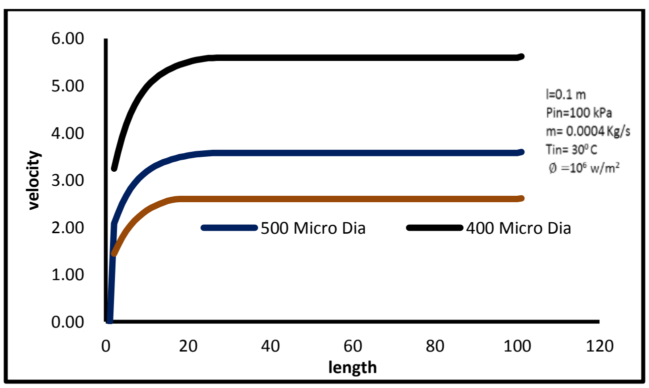

Figure 8 velocity profile at center line in circular micro channel with different diameter

Conclusions

The influence characteristics and mechanisms of circular micro-channel were analyzed numerically. The numerical simulation has been performed in the ANSYS 15.0. Five indicators, including temperature, enthalpy, velocity, Reynolds number and effect of diameter on the enthalpy and velocity, that were commonly used, and proposed in this study, were evaluated. Following conclusions were drawn:

- Temperature is linearly increased along with axis length of micro channel.
- Velocity and Reynolds number is initially increased along with axis length of micro channel then it's becoming constant.
- The bulk enthalpy of the micro-channel is increases with increases in the diameter.
- The average velocity of the micro-channel is increases with decreases in the diameter.

References:

[1] S.K.Rai, Review of recent applications of micro channel in MEMS devices, International Journal of Applied Engineering Research ,13.9, 64-69, (2018).

[2] S.K. Rai and G. Dutta, A Review of Recent Applications of Supercritical Fluid in Natural Circulation Loops for Nuclear Reactor. International Journal of Applied Engineering Research ,23 ,195-204 (2018)

[3] H. K. GUPTA, S. Kumar, and T. Sheorey, Boiling flow heat transfer in microchannel: Experimental and numerical investigation", In Proceedings of the 23rd National Heat and Mass Transfer Conference and 1st International ISHMT-ASTFE Heat and Mass Transfer Conference IHMTC-2015, 1-8, Trivendram, INDIA, (2015)

[4] D.B. Tuckerman, R.F. Pease, High performance heat sinking for VLSI, IEEE Electron. Devices Lett. (1981) EDL 2 pp.126–129.

[5] P.S Lee, S.V Garimella, D. Liu, Investigation of heat transfer in rectangular microchannels, International journal of heat and mass transfer, 48 (2005) 1688–1704.

[6] W. Qu, I. Mudawar, Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink, International Journal of Heat and Mass Transfer 45 (2002) 2549–256.

[7] P. Gunnasegaran, H.A. Mohammed, N.H Shauib, R.Saidur, The effects of geometrical parameters on heat transfer characteristics of microchannels heat sink with different shapes, International communications in heat and mass transfer 37 (2010) 1078-1086.

[8] H.A. Mohammed, P. Gunnasegaran, N.H Shauib, Influence of channel shape on the thermal and hydraulic performance of microchannel heat sink, International communications in heat and mass transfer 38 (2011) 474-480.

[9] N. Ghazali-Mohd, O. J. Taek, N. B. Chien, C. Kwang, R. Ahmad, Compariosn of the optimized thermal performance of square and circular ammonia-cooled microchannel heat sink with genetic algorithm, Energy conversion and management (2015).

[10] S.K. Rai, P. Kumar, and V. Panwar, Numerical analysis of influence of geometry and operating parameters on Ledinegg and dynamic instability on supercritical water natural circulation LOOP, NUCLEAR Engineering and Design, 369,110830 (2020)

[11] S. K Rai, P. Kumar, V. Panwar, Mathematical and numerical investigation of Ledinegg flow excursion and dynamic instability of natural circulation loop at supercritical condition, Annals of Nuclear Energy, 155, 108129 (2021)

[12] S.K. Rai, P. Kumar and V. Panwar, Numerical investigation of steady state characteristics and stability of supercritical water natural circulation loop of a heater and cooler arrangements, Nuclear Engineering and Technology, 2022

[13] L. Chai, S.A. Tassou, Effect of cross-section geometry on the thermohydraulic characteristics of supercritical Co2 in minichannel, Energy Procedia 161 (2019) 446–453.

[14] S.E. Ghasemi, A.A. Ranjbar, M.J. Hosseini, Experimental and numerical investigation of circular minichannel heat sinks with various hydraulic diameter for electronic cooling applications, Microelectronics Reliability 73 (2017) 97-105.

[15] G.V. Kewalramani, G. Hedau, S.K. Saha, A. Agrawal, Empirical correlation of laminar forced convective flow in trapezoidal microchannel based on experimental and 3D numerical study, International Journal of thermal science 142 (2019) 422-433.

[16] L. Su, Z. Duan, B. He, H. Ma, G. Ding, Laminar Flow and Heat transfer in the entrance region of elliptical minichannels, International Journal of Heat and Transfer 145 (2019) 118717.

[17] F.S. Moghanlou, A.S. Khorrami, E. Esmaeilzadeh, H. Aminfar, Experimental study on electrohydrodynamically induced heat transfer enhancement in a minichannel, Experimental Thermal and Fluid Science 59 (2014) 24–31.

[18] C.J. Ho, P.C. Chang, W.M. Yan, P. Amani, Thermal and hydrodynamic characteristics of divergent rectangular minichannel heat sinks, International Journal of heat and mass transfer 122 (2018) 264-274.

[19] S.G. Khandlikar, Single-Phase Liquid Flow in Minichannels and Microchannels, Elsevier publication, 2014.

[20] Tuckerman, D. B., Pease, R. F.: High Performance Heat Sinking for VLSI, IEEE Electronic Device Letters, EDL-2(1981).

[21] Kandlikar S. G., Colin S., Peles Y., Garimella S., Pease R.F., Brandner J.J., Tuckerman, Heat transfer in microchannel2012 status and research needs, ASMEJ of Heat Transfer,Sept., Vol.135, (2013).

[22] Gaku HAYASE, Development of Micro Channel Heat Exchanger for Residential Air-Conditioners, International Refrigeration and Air Conditioning Conference. Paper 1566.

[23] Jaeseon Lee, Issam Mudawar, Two-phase flow in highheat-flux micro-channel heat sink for refrigeration cooling applications: PartI––pressure drop characteristics, Elsevier, International Journal of Heat and Mass Transfer 48 (2005) 928–940.

[24] Yanhui Hana, Yan Liua, Ming Liaa , Jin Huanga, A review of development of micro-channel heat exchanger applied in air-conditioning system, Energy Procedia 14 (2012) 148 –153

[25] Ahmet Selim DALKILIC Mohamed M. AWAD, and Somchai WONGWISES, Air-side performance of a micro-channel heat Exchanger in wet surface conditions, THERMAL SCIENCE: Year 2017, Vol. 21, No. 1A, pp. 375-385

[26] Huize Li, Pega Hrnjak, Quantification ofliquid refrigerant distribution in parallel flow microchannel heat exchanger using infrared thermography, Applied Thermal Engineering 78 (2015) 410e418

[27] KAMATA, KIM, FUJINO, 2012, PROCEEDINGS OF THE 2012 JSRAE ANNUAL CONFERENCE, SAPPORO, PP37-40

[28] E. S. CHO, J. W. CHOI, J. S. YOON AND M. S. KIM, "MODELING AND SIMULATION ON THE MASS FLOW DISTRIBUTION IN MICROCHANNEL HEAT SINKS WITH NON-UNIFORM HEAT FLUX CONDITIONS,"INTERNATIONAL JOURNAL OFHEAT AND MASSTRANSFER, VOL. 53, P. 1341–1348, 2010.

[29] Y.-T. MU, L. CHEN, Y.-L. HE AND W.-Q. TAO, "NUMERICAL STUDY ON TEMPERATURE UNIFORMITY IN A NOVEL MINI-CHANNEL HEAT SINK WITH DIFFERENT FLOW FIELD CONFIGURATIONS, "INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER, VOL. 85, PP. 147-157, 2015.