Bilge International Journal of Science and Technology Research

Web : http://dergipark.gov.tr/bilgesci - **E-mail:** kutbilgescience@gmail.com

Received: 18.11.2019
Accepted: 25.12.2019
DOI: 10.30516/bilgesci.648096

ISSN: 2651-401X
e-ISSN: 2651-4028
3(Special Issue), 45-57, 2019

Numerical Simulation of Annular Flow boiling in Millimeter-scale Channels and Investigation of Design Parameters Using Taguchi Method

Aliihsan Koca[1]*, Mansour Nasiri Khalajı[2]

Abstract: As the technology progresses, the electronic components become smaller and at the same time continue to produce more heat, and therefore development of new high heat-flux cooling technologies have become obligatory. The mini and millimeter-scale phase change cooling systems, which have a reduced size and a large surface area where heat transfer can take place, have become an integral part of advanced cooling systems. When comparing phase-change cooling systems with other cooling systems, a relatively low flow rate of very high evaporation heat, which is associated with the phase change for most fluids, allows large amounts of heat to dissipate with flow boiling and substantially solves the many problems. The two-phase cooling technologies used for critical applications include; heat pipes, loop heat pipes and capillary pumped loops which are all passive hence very reliable solutions relying on only capillary effects. Though this passive device cannot meet future high cooling demands because of the limitations of the capillary pumping in terms of heat flux, transport distance and multiple heat source capabilities. On the other hand, in boiling and condensing flows functionality problems arise since at the micrometer and millimeter-scale, shear/pressure forces dominate over gravitational forces and cause thermally hydro-dynamically ineffective/problematic liquid-vapor configurations – such as plug/slugs flow regimes. For this reason, to overcome the requirement of large amounts of heat transfer from limited spaces and resolving the above problem, novel millimeter-scale phase-change devices should be developed. In this study, for the design of millimeter-scale boilers a 3D Ansys-Fluent© simulation model was developed and numerical simulations were conducted for two different cooling fluids (water and FC-72), different mass flow rates and two different channel heights. Moreover, to examine the simulation results Taguchi method was used. In order to realize thin film annular flow over the boiler surface, employed specific boundary conditions in the 3D simulation model were obtained by means of one dimensional Matlab© simulation code. By means of utilizing the evaluated numerical results, distribution of heat transfer coefficient, vapor quality and pressure drop over the heat transfer surfaces were reported.

Keywords: Heat Transfer, Boiling flow, Phase Change, Computational Fluid Dynamics, Taguchi Method.

1. Introduction

The rapid improvements in the performance of miniature and electronic devices have resulted in a significant challenge in the thermal management of these devices. The heat dissipation from these high-performance and compact devices has reached great values, and as a result traditional cooling methods such as air cooling have become ineffective. The heat transfer coefficient in applications such as high-power lasers, microwave devices and radars is about 10 MW/m^2 (Lee and Mudawar, 2008). However, overpressure drop and uneven temperature distribution can be accompanied by electronic chips. In contrast, flow boiling in mini and millimeter channel heat sinks

[1]Fatih Sultan Mehmet Vakif University, Faculty of Engineering, 34445, Istanbul, Turkey
[2]Ataturk University, Faculty of Engineering, 25240, Erzurum/Turkey
* Corresponding author: * ihsankoca@hotmail.com

Citation (Atıf): Koca, A., Khalaji, M.N. (2019). Numerical Simulation of Annular Flow boiling in Millimeter-scale Channels and Investigation of Design Parameters Using Taguchi Method. Bilge International Journal of Science and Technology Research, 3(Special Issue): 45-57.

can achieve higher heat transfer rates, better axial temperature homogeneity, lower mass flow rates, and lower pressure differential or less pumping power than single-phase flow (Consolini and Thome, 2009; Karayiannis et al., 2010).

The increasing development of electronical systems has become an increasing trend among the separate branches of the Defense industry (Park and Vallury, 2006; Park and Jaura, 2002; Kuszewski and Zerby 2012; Ponnappan et al., 2002; Park and Zuo, 2004).

In the defense industry of many developed countries, research has been conducted to increase the power density while reducing the size of the hybrid electric vehicle, electrical electronic subsystems, and the component parts to increase the component power that supports this strategy (Moore, 1993; Urciuoli et al., 2012). However, the new generation of electronic systems produces multiple kilowatt waste heat with heat fluxes at a predicted future device level up to 1000 W/cm² (Lee and Mudawar, 2009). To achieve this, innovative condensers and boilers operating in the cyclic regime and interrupted by the negligible effect of gravity must be developed (Kivisalu et al., 2014). The advantages of operating these capacitors in pulsatile mode are also discussed. Such millimeter-scale capacitors (Kivisalu et al., 2014) are of great value in the design of new generation space-based thermal systems and gravity insensitive aircraft-based systems (including avionics cooling).

The application of various micro and mini channel designs has started to gain considerable importance (Tuckerman and Pease, 1981; Sullivan, et al., 1992; Hall and Mudawar, 1995). Regardless of a particular heat sink configuration, critical cooling options carried out beyond a laboratory environment are based on almost single-phase liquid flow designs (Mudawar, 2009). But with the ever-increasing electronic heat flows and the installation of single-phase thermal management problems, cooling schemes using liquid-vapor phase change (two-phase cooling) have been examined as a new generation cooler superior to single-phase cooling systems that are practical and cost-conscious. In many studies, it has already well-known two-phase flow technologies as an appropriate solution to meet the stringent demands for high cooling needs requirements (Kuszewski and Zerby 2012; Ponnappan et al., 2002). In the two-phase cooling cycle, when it exceeds the boiling point, a portion of it is obtained in a gas/liquid mixture and at least a portion of the liquid is converted into vapor upon heating. This generally means that the temperature of the heat-collecting surface is called the saturation temperature of the liquid or boiling and the use of a cooling fluid. At this temperature, the vapor pressure of the liquid is equal to the ambient pressure, thus causing vapor bubbles to form, grow and eventually detach from the surface at the solid and liquid interface. As shown in Figure 1, the heat transfer coefficients obtained from the two-phase flow may have a greater heat transfer than equivalent single-phase forced or natural heat convection (Mudawar, 2001).

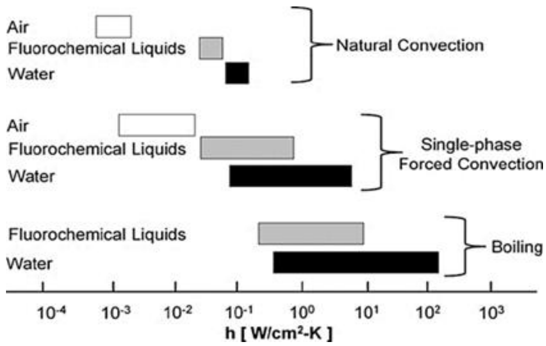


Figure 1. Heat transfer coefficients for various fluids (Mudawar, 2001).

Two-phase mini and millimeter channel systems have received great interest as a method for electronic cooling with high heat flux. Although it has been shown that flow in the large surface area of the microchannels increases the heat transfer coefficient in single-phase flow (Tuckerman and Pease, 1981; Phillips, 1990). Additional gains in heat transfer coefficient can be obtained by allowing the liquid to evaporate along the mini and millimeter channel walls. For these reasons, recent research has focused on two-phase cooling and mini and millimeter channel flow boiling. The controlled heat transfer mechanism in the mini and millimeter channels is thought to be the evaporation of the thin liquid film around the bubbles in the micro-channels (Chen, et al., 2013; Mersen, 2017). There are several general literature reviews of one-dimensional and two-dimensional FC-72 flow boiling and heat transfer, depending on channel geometry (Pereira et al., 2017; Ansys, 2017; Kivisalu et al., 2014; Naik, et al.2014).

In this study, we will examine the numerical heat transfer with different ducts at different heights, refrigerants with different properties, different flow rates, different vapor qualities, fluid pressures and different heat fluxes with Taguchi method.

2. Material and Method

2.1. Taguchi method

The Taguchi method is an experimental design method that tries to minimize the number of experiments by selecting the most optimal combination of levels of factors that can be controlled against uncontrollable factors that consist of variables and levels before conducting experiments (Canıyılmaz, 2001; Ömeroğlu, 2018).

In this method, one of the ranking method, observation method, variance method, analysis of factor effects, column differences method and graphical representation methods are applied to determine factor levels (Ross, 1989). In another study (Caliskan, et al., 2015), using the Taguchi method, Reynolds's number and jet-plate distances performed heat transfer measurements on the surface for a sequential jet array.

In this study, by means of using Taguchi experimental design method, optimum levels of parametric values were applied in order to examine the in heat transfer characteristics in a millimeter channel.

Taguchi analysis was carried out by using Minitab 18.0©, which has a capability to evaluate the statistical functions. In the Minitab software, orthogonal array (OA) designs for the array design, whose degrees of freedom should be greater than or at least equal to those of the design parameters, generated. Therefore, L8 corresponds to the design considered and the factors are shown in Table 1. The main target is to increase the bubbles and therefore maximum heat transfer. The performance statistics were preferred as the optimization measure and implemented for the “large is better” condition and determined by means of the below equation.

$$Z_L = -10\log\left(\frac{1}{n}\sum_{i=0}^{n}\frac{1}{Y_i^2}\right) \qquad (1)$$

Where, Y is the performance value of the experiments and Z shows the performance statistics, while n is the repetition number in the experiments.

Table 1. Parameters and their values corresponding to their levels

Parameters	Levels	
Channel height	5mm	8mm
Fluid Material	Water	FC-72
Heat flux	107 w/m^2	200 w/m^2
Pressure	1.5 bar	3 bar
Total Mass flow rate	140 kg/m^{2}s	200 kg/m^{2}s
Steam quality	0.61	0.67

In the test plan, optimum operating conditions for optimum values of parameters affecting system performance may not be always available. Ideal operating conditions which correspond to performance could be evaluated by means using the Orthogonal Array (OA) balanced character.

In Table 2 and Table 3, the contributing ratios of all factors on the performance are specified based on the SNR values. In the tables ideal combination of the important parameters can be estimated. For implementing the reproducibility approval procedure, the deviation of the SNR between the current and ideal conditions after the first prediction was attained. From the SNR deviations, the ideal parameters estimated from the parameter design can be validated.

The levels of the process parameters and factors are given in Table 2, and Table 3 shows the orthogonal sequence L8 of the desired experimental design.

Table 2. Experimental L8 (2 ^ 6) orthogonal sequence and SNR values plan

Ex no	Channel height (A)	Fluid Material (B)	Heat flux (C)	Pressure (D)	Mass flow rate (E)	Steam quality (F)
1	1	1	1	1	1	1
2	1	1	1	2	2	2
3	1	2	2	1	1	2
4	1	2	2	2	2	1
5	2	1	2	1	2	1
6	2	1	2	2	1	2
7	2	2	1	1	2	2
8	2	2	1	2	1	1

Table 3. Numerical plan of L8 (2 ^ 6) orthogonal sequence and SNR values

Plane Vapor Quality	SNR1	Outlet Vapor Quality	SNR2	Plane Thermal Conductivity	SNR3
0.67	-3.4785	0.65	-3.7417	0.21	-13.5556
0.72	-2.8533	0.7	-3.0980	0.18	-14.8945
0.69	-3.2230	0.69	-3.2230	16.5	24.3496
0.64	-3.8764	0.66	-3.6091	21.12	26.4938
0.64	-3.8764	0.65	-3.7417	0.23	-12.7654
0.69	-3.2230	0.68	-3.3498	0.21	-13.5556
0.69	-3.2230	0.68	-3.3498	22.18	26.1945
0.62	-4.1521	0.63	-4.0131	24	27.6042

Tables 4, 5 and 6 and Figures 2, 3 and 4 show the S/N or signal to noise ratio and variable level. High delta refers to the high S/N ratio variation of the design parameter. Sequence refers to the surface area, which is the most important design parameter.

Table 4. Contribution ratio and factorial effect for vapor volume fraction

Response Table for Signal to Noise Ratios for plane Vapor friction						
Level	A	B	C	D	E	F
1	-3.36	-3.36	-3.43	-3.45	-3.52	-3.85
2	-3.62	-3.62	-3.55	-3.53	-3.46	-3.13
Delta	0.261	0.261	0.12	0.07	0.06	0.71
Rank	3	2	4	5	6	1

As shown in the figure 2, 3 and 4, the millimeter channel heat exchanger is open with all three effective parameters, increasing the channel height, the type of fluid flowing, and the mass efficiency as well as increasing efficiency.

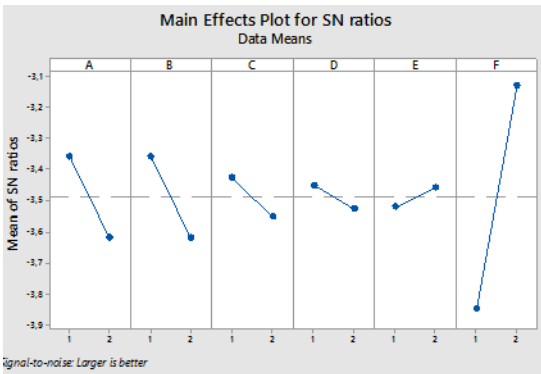


Figure 2. The Effect of the Parameter On Performance Statistics (For vapor quality in the plate).

Table 5. Factorial effect and contribution ratio for projected outlet vapor volume fraction

Response Table for Signal to Noise Ratios for plane Vapor friction						
Level	A	B	C	D	E	F
1	-3.42	-3.48	-3.55	-3.51	-3.58	-3.78
2	-3.61	-3.55	-3.48	-3.52	-3.45	-3.25
Delta	0.20	0.07	0.07	0.003	0.132	0.52
Rank	2	5	4	6	3	1

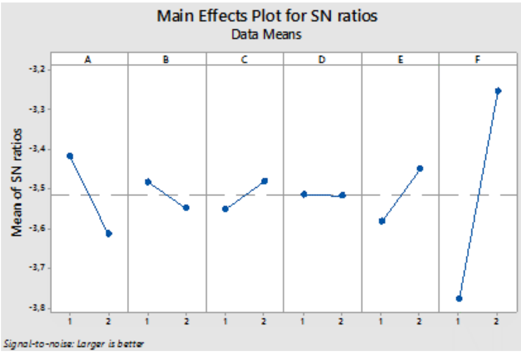


Figure 3. The Effect of the Parameter on Performance Statistics (for vapor quality in outlet).

Table 6. Contribution ratio and factorial effect for thermal conductivity

Response Table for Signal to Noise Ratios for plane Vapor friction						
Level	A	B	C	D	E	F
1	5.598	-13.7	6.52	6.24	6.21	6.94
2	7.051	26.34	6.13	6.41	6.44	5.70
Delta	1.452	40.03	0.39	0.17	0.23	1.24
Rank	2	1	4	6	5	3

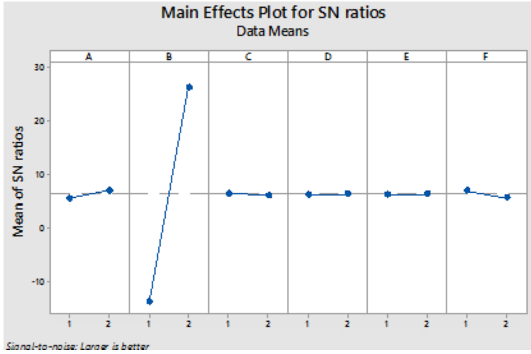


Figure 4. The effect of the parameter on performance statistics (for thermal conductivity in plate).

Taguchi analysis is also useful for predicting the best case from selected experimental cases. In this study, a response named outlet temperature was selected and the S/N ratio results discussed in the previous sections. Figures 2, 3 and 4 are 5 millimeters away from the x, y plane and the z plane, respectively. The steam quality value shows the average ratio and S/N ratio results for responses with the value of the steam quality at the outlet and the thermal conductivity value. As it can be seen from the tables and figures, for both fluid flow steam quality plays an important role in the measurement of vapor volume fraction and fluid material is considered as one of the most important and effective parameters in heat transfer coefficient.

2.1. Flow boiling in mini and millimeter Channels

In two-phase heat transfer systems, it can operate at lower flow rates and lower pumping power than single-phase forced convection heat exchangers based on latent heat of the working fluid. A single-phase liquid system with a temperature increase limited to 25 ° C will require approximately seven times the flow rate of a two-phase system to reject the same amount of heat (Agostini, 2007; Hannemann, et al., 2004; Marcinichen and Thome, 2010; Pan, et al. 2015). It also ensures lower flow rates and smaller thermal management systems (Mudawar, 2001; Willingham and Mudawar, 1992). In a study related to this, (Mishima and Hibiki, 1996) examined the air/water flow of the vertical and horizontally directed capillary tubes connected to the closed rings and found that these same capillary forces affect the bubble dynamics and the flow regime of the air/water mixture.

Based on the analysis of data from various sources (Tran, et al.1996; Lazarek and Black, 1982; Cooper, 1984; Cooper, 1989; Liu and Winterton, 1991), they proved that the micro-regime had begun if the number of limitations was greater than 0.5 in another study (Kuznetsov, 2013), when the width of the channel falls below the capillary constant (δc), it points to a similar criterion at which the mini and millimeter regime begin:

$$\delta_c = \sqrt{\frac{2\sigma}{g(\rho_1 - \rho_2)}} \qquad (2)$$

They investigated three micro, mini and macro regimes on Bond number (Bo), which is also related to surface tension, gravity and hydraulic diameter (Cheng, et al. 2007).

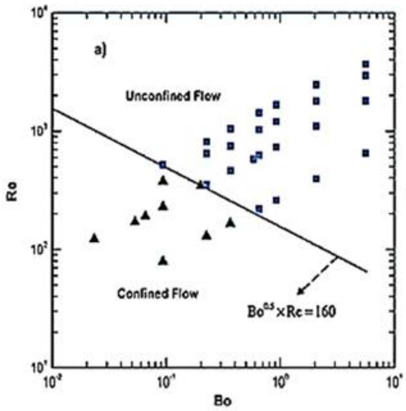
$$Bo = \frac{g(\rho_1 - \rho_2)D_h^2}{\sigma} \qquad (3)$$

They also investigated that the gravitational effects of the mini and millimeter channels and that the Bond number is less than 0.05 can be overlooked in this region, but in small channels the Bond number (Bo) is between 5 and 30 and proved that the surface tension and gravitational effects are small but still present. These results are shown in Figure 2a (Harirchian and Garimella, 2009a; Harirchian and Garimella, 2009b; Thome, et al., 2013). They proposed the convective limiting number as a combination of the Bond number and the Reynolds number, and this is the correct method to define the flow limitation.

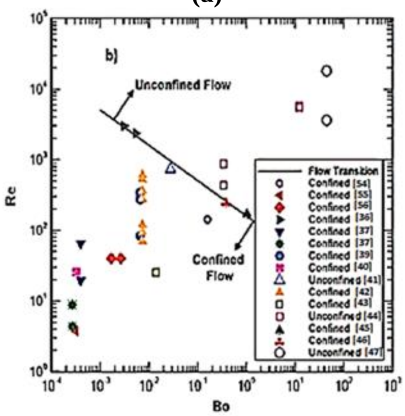
$$Bo^{1/2} \times Re = \frac{GL_c^2}{\mu_1}\left(\frac{g(\rho_1 - \rho_2)D_h^2}{\sigma}\right)^{1/2} \qquad (4)$$

Here the viscosity (μ) is the mass flux (G) and the characteristic length (Lc) is the square root of the cross-sectional area of the channel as opposed to the normal hydraulic diameter. Two simulation tools and mathematical models for the definition of the above were proposed in separate articles (Narain, et al. 2004; Liang, et al., 2004).

The limitation is a feature of mini and millimeter channel flow, which can be regarded as decisive criteria between mini, millimeter or macro. In many studies, the authors compared these transition criteria with other experimental data using different fluids, and all made good estimates. (Figure 2b). In Figure 2-b, the researchers emphasize that the cross-sectional area rather than the hydraulic diameter or aspect ratio is important in this determination. Although there is no clear consensus as to what distinguishes the mini and millimeter channel from a conventional channel, it is clear that there are significant differences between the two dimensions.



(a)



(b)

Figure 5. (a) A combination of the convective limiting number, the Bond number and the Reynolds number (Harirchian and Garimella, 2009b; Thome, et al., 2013). (b) The transition from various other studies to the unrestricted flow

Three boundary conditions are required to fully uncover the mini and millimeter channel problem and solve the equations. Each channel has its boundary conditions are set in Fluent. Boundary conditions include:

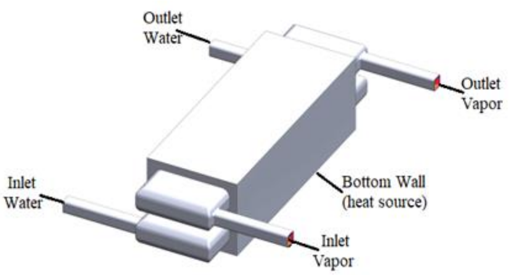
• Mass flow rate (depends on the input limit of mini and spindle channel)
• Output pressure (depends on mini and spindle channel output limit)
• Wall temperature (depends on mini and spindle channel wall limit)

Basic equations are discretized using finite difference schemes.

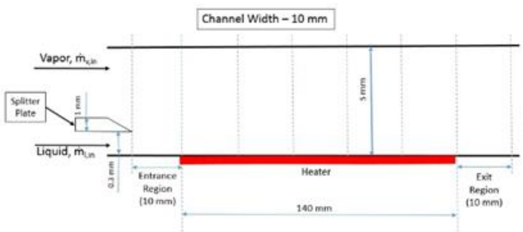
We first designed the geometry using drawing programs such as SolidWorks. Typically, the CAD geometry consists solely of the solid body structure from which the design is produced.

Figure 6a shows a millimeter channel heat exchanger which is used as the CFD model. In the model, two-phase liquid fields were formed along with the heat-exchanger by means of the thermal energy supplied from the bottom wall. Each phases (liquid and vapor) enter the system from different inlets and exit from two different outlets.

Figure 6b shows a schematic dual phase heat exchanger of the same millimeter channel which were used to develop simple dual input and dual output.



(a)



(b)

Figure 6. Two-input and two-output dual-phase heat exchanger (a) 3D view (b) cross-section view (Soroush, 2019)

In further applications of CFD, geometric simplifications were made during geometry generation by methods such as 3D or symmetrical to reduce the complexity of areas that do not affect the physics of the overall solution. For conjugate heat transfer problems, such as heat exchanger modelling, it is important to construct the geometry of the liquid areas in such a way that the solid boundaries in contact with the liquid are perfect and matched. The stability of the solutions were obtained from two different methods (1-D and 2-D) is given in (Naik, et al. 2015) and the stability of gravity-driven and shear-driven flows is given in (Naik, et al. 2015), respectively.

After the geometry is created for both the gas and liquid areas, the calculation area in which the CFD simulation is solved was created. By using ANSYS Mesher©, the best mesh was produced using different mesh configurations. For conjugate heat transfer problems, it is common practice to connect solid and fluid areas to mesh using appropriate methods, and to provide a unity between areas (providing a one-to-one mesh face). Conformal meshes eliminate any interpolation found in incompatible meshes, providing accuracy for improved mesh production. In all simulation cases, the solution is highly dependent on the mesh resolution. To understand this situation, it is useful to complete a network convergence study and a mesh validation test must be performed in this. For this reason, a mesh independency study was performed.

Then, for an optimal solution, simulations were carried out under predefined initial and boundary conditions, which depend on the desired operating conditions of the design. Main boundary conditions for heat exchangers are energy operating conditions, such as flow conditions passing through each fluid area, such as mass flow rates and pressure properties, or initial temperatures of liquids entering the heat exchanger. This study only deals with steady-state cases, but it is necessary to establish initial conditions for transient cases in other CFD problems.

Simulation with appropriate mesh and initial and boundary conditions can now be approached for simulation resolution, with conditions set in the CFD solvent. Under start-up conditions, a desired mass flow of liquid at a given temperature can be performed in many ways using manual initiation or automatic methods. With the combined algorithm, the iterative decoding process provides the temperature, speed and pressure profiles for each of the channel flows by solving the annular flow model using the current conditions of the CFD simulation and the solution was initiated. The linked algorithm is iterative and each of the solutions for the model and CFD simulation will be interconnected, and once a repetition is complete, the channel outlet pressure, wall temperature profile, and mass flow rates through the channels are updated according to the flow model. ANSYS Fluent was used to perform numerical functions and the liquid-vapor interface in this simulation was solved by accepting a finite volume (VOF) model. In this model, by solving an additional continuity-like equation for the volume fraction, it is assumed and realized that the two-phases are incompressible and do not penetrate each other. The sum of the volume fractions of the two-phases in each cell is combined and the properties of a single liquid are calculated based on the volume weight fraction of each phase in the cell. In this study, a group of mathematical models, conservation laws for mass, momentum and energy were solved.

In the current analysis study, the VOF model was used to simulate multiphase flow, while gravitational acceleration was given on the minus y axis and its value was taken as 9.813 m/s2. Furthermore, water and FC-72 were used as the working fluid, and the thermophysical properties of the fluids at the saturation temperature corresponding to the fluid 1bar are shown in Table 1 from the Fluent© substance database.

The liquid water was selected as the secondary phase (liquid phase) and, due to the dynamic behaviour of the two-phase flow, a transient solution with a period time of 0.001 s was used for the all cases. A combination of the SIMPLE algorithm for pressure-speed coupling and the calculation of momentum and energy and the standard k-ε model were used to model turbulence. Table 7 shows the thermodynamic properties of both fluids and the calculations were made by means using the default data in this table.

Table 7. Properties of working fluids used in simulation

Features	Water (L)	Water (V)	FC-72 (L)	FC-72 (V)
Density (kg/m^3)	1000	0.5542	1674.75	13.01
Cp (J/kg*K)	4182	2014	1052.85	1100.24
Thermal Conductivity (W/m*K)	0.6	0.0261	0.05725	0.0536
Viscosity (kg/m*s)	0.009	13e-6	0.00064	0.00043
Molecular Weight (kg/kmol)	18.01	18.0152	340	340
Reference Temperature (K)	298.15	298.15	329.15	329.15

In order to reduce the simulation time of the boiling process, the initial temperature of both the boiling surface and the liquid was selected above the boiling point. As the coolant fluid, perfluorohexane, a Fluorinert™, commercially as FC-72 was used. The basic physical properties of this dielectric fluid are reported in Table 1. FC-72 is thermally and chemically stable, compatible with sensitive materials, flammable, non-toxic, colorless and has no ozone depletion potential. This combination of features makes the FC-72 particularly suitable for applications such as heat sinks for electronic components in combination with low viscosity. It should be also noted that the latent heat of the FC-72 is significantly higher (88 kJ/kg) than the specific heat capacity (1.1 kJ/kgK).

3. Results and Discussion

In the project a one-dimensional Matlab© simulation code was developed using existing theories/correlations in the literature in order to design a boiler. Using this simulation code, evaporator designs can be realized for different fields of application and operating conditions. Afterwards 3D simulation studies were carried out for the boiler geometry which was designed by means of using the developed one-dimensional Matlab© simulation code. In this study, as refrigerant; FC-72 (Fluorinert™) and water were used. then, using the Taguchi (L8) method, after eight analyzes, as shown in T0able 3, the S/N numbers were calculated and tabulated, and finally the heat transfer coefficient and steam quality for both fluids were calculated and compared to the values obtained in the Matlab© code.

By the proposed new method (continuous steam recirculation), the thin film was provided for different thermal boundary conditions of the continuous flow regime, eliminating hydrodynamic and other thermally inefficient flow regimes.

Figures 7 and 9 illustrate their effect on heat transfer coefficients and steam quality for each experiment. The heat transfer coefficients of the 8 mm milichannel are approximately 12% lower than that of the 5 mm millimeter channel. This may be due to an increase in perimeter which contributes to the effective distribution of the heat load to each channel (5 and 8 mm). In addition, the heat transfer coefficients of steam quality in the 8 mm millimeter channel are smaller than those in the 5 mm millimeter channel. As a result, the height of the channel causes an adverse effect.

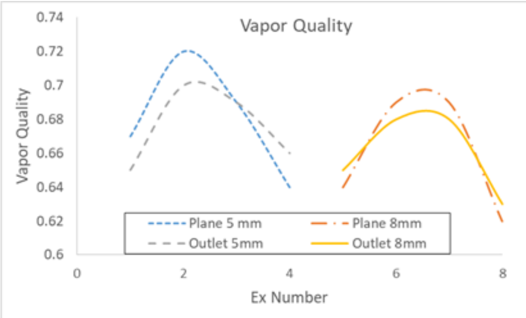


Figure 7. Vapor quality value for each experiment

Figures of 8, 11 and 12, illustrates the calculated thermal conductivity values of the fluids for different cases.

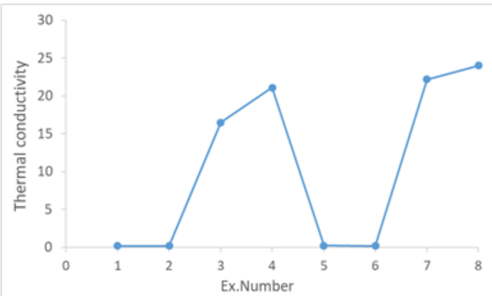


Figure 8. Thermal conductivity value for each experiment

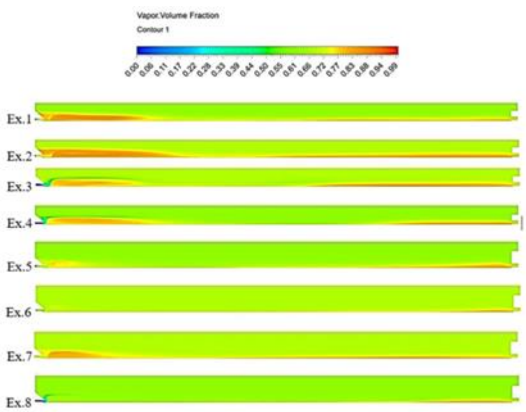


Figure 9. Vapor volume fraction contours for each experiment

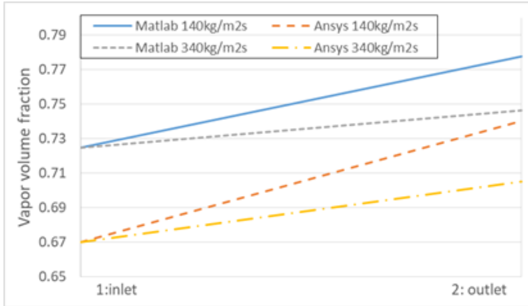


Figure 10. Vapor volume fraction value for water each experiment between inlet and outlet

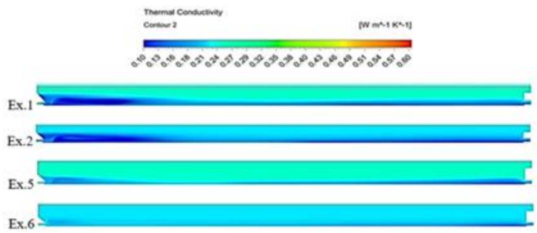


Figure 11. Thermal conductivity contours for water each experiment

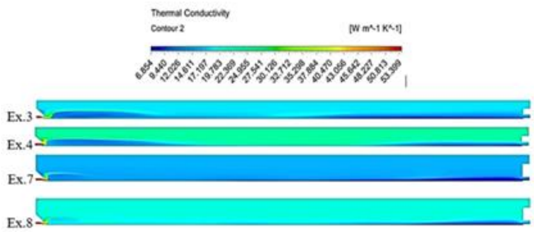


Figure 12. Thermal conductivity contours for FC-72 each experiment

Figures 13 and 14 show the velocity vector of the millimeter channels of fluids near the inlet having both heights (5mm and 8mm). In the 5mm channel, the steam flow enters having higher velocity than the 8mm channel, which can cause a high pressure difference.

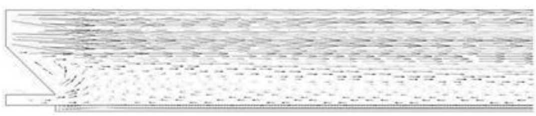


Figure 13. Fluid flow vector for 5 mm channel

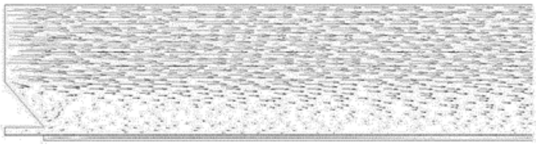


Figure 14. Fluid flow vector for 8 mm channel

4. Conclusions

Numerical research has been conducted to investigate the flow boiling of water vapor and FC-72 at different flow rates, heat fluxes, pressures, vapor qualities. The effect these parameters on heat system performance were discussed. Finally, the main results are plotted as follows:

• Vapor quality increases with the increase of steam flow and hence the Reynolds number, but as millimeter channel height increases, the heat transfer coefficient decreases.

• Depending on the hydraulic diameter of the mini channel, the Reynolds number of high inlet vapor and the boiling in the millimeter channel for the external heat flux will be greater than necessary conditions.

• Wall temperature decreases along the flow direction to increase thin film thicknesses. This may result in increased thermal resistance in the wall. However, in order to avoid dry-out instability at the outlet, film thickness was maintained at certain values (minimum 30 μm).

Acknowledgements

This work was supported by the Scientific and Technological Research council of Turkey (TUBITAK). The study was a part of the TUBITAK 3501 project with the number of 118M457.

References

Agostini, B., et al. (2007). State of the Art of High Heat Flux Cooling Technologies. Heat Transf. Eng., 28(4), pp. 258-281.

Ansys, (2017, October.). Electromagnetic simulation products [Online]. Avaible: http://www.ansys.com/products/electronics

Bevis, T.A. (2016). High Heat Flux Phase Change Thermal Management of Laser Diode Arrays. Colorado State University, PhD Thesis.

Caliskan S., Nasiri Khalaji M., Baskaya S., Kotcioglu I. (2015). Design Analysis of Impinging Jet Array Heat Transfer from a Surface with V-shaped and Convergent-divergent Ribs by Taguchi Method. Heat Transfer Engineering 37(15), pp1252-1266.

Cheng, P., Wu, H.Y., and Hong, F.J. (2007). Phase-Change Heat Transfer in Microsystems. J. Heat Transfer, 129(2), p. 101.

Canıyılmaz, E. (2001). Kalite Geliştirmede Taguchi Metodu ve Bir Uygulama, Yüksek Lisans Tezi, Gazi Üniversitesi.

Chen, Z. et al. (2013). Development of a 1200 V, 120 A SiC MOSFET Module for High-temperature and High-frequency Applications. In the 1st IEEE Workshop on Wide Bandgap Power Devices and Applications, pp.52-59.

Cooper, M.G. (1984). Saturated Nucleate Pool Boiling – a Simple Correlation. Proc. Of the 1st UK National Heat Transfer Conference, IChemE Symposium, pp. 785-793.

Cooper, M.G. (1989). Flow Boiling-the ‘apparently Nucleate Regime. Int. J. Heat Mass Transf., 32(3). 459-464.

Consolini, L., Thome J.R. (2009). Microchannel Flow Boiling Heat Transfer of R134a. R236fa and R245fa, Microfluidics and Nanofluid, 6 731-746.

Hall, D.D., Mudawar, I. (1995). Experimental and Numerical Study of Quenching Complex-Shaped Metallic Alloys with Multiple, Overlapping Sprays. International Journal of Heat and Mass Transfer, 38, 1201-1216.

Hannemann, R., Joseph, M., Pitasi, M. (2004). Pumped Liquid Multiphase Cooling. IMECE, pp. 3-7.

Garimella, S.V, Yeh, L., and Persoons, T. (2012). Thermal Management Challenges in Telecommunication Systems and Data Centers. IEEE Trans. Components, 2(8), pp. 1307- 1316.

Garimella, S.V, Persoons, T., Weibel, J., Yeh, L.T. (2013). Technological Drivers in Data Centers and Telecom Systems: Multiscale Thermal, Electrical, and Energy Management. Appl. Energy, 107, pp. 66-80.

Harirchian, T., and Garimella, S.V. (2009a). The Critical Role of Channel Cross-Sectional Area in Microchannel Flow Boiling Heat Transfer. Int. J. Multiph. Flow, 35, pp. 904-913.

Harirchian, T., and Garimella, S.V. (2009b). Effects of Channel Dimension, Heat Flux, and Mass Flux on Flow Boiling Regimes in Microchannels. Int. J. Multiph. Flow, 35(4), pp. 349-362.

Karayiannis, T.G., et al. (2010). Flow pattern and heat transfer for flow boiling in small to micro diameter tubes. Heat Transfer Engineering, 31, 257-275.

Karayiannis, T.G., and Mahmoud, M.M. (2017). Flow Boiling in Microchannels: Fundamentals and Applications. Appl. Therm. Eng., 115, pp.1372-1397.

Kandlikar, S.G., and Grande, W.J., (2003). Evolution of Microchannel Flow Passages-Thermohydraulic Performance and Fabrication Technology. Heat Transf. Eng., 24(1), pp. 3-17.

Kandlikar, S.G. (2012). History, Advances, and Challenges in Liquid Flow and Flow Boiling Heat Transfer in Microchannels: A Critical Review. J. Heat Transfer, 134 (3).

Kew, P. A., and Cornwell, K. (1997). Correlations for the Prediction of Boiling Heat Transfer in Small Diameter Channels. Appl. Therm. Eng., 17, pp. 705-715.

Kivisalu, M.T., Gorgitrattanagul, P., and Narain, A. (2014). Results for high heat-flux flow realizations in innovative operations of milli-meter scale condensers and boilers. International Journal of Heat and Mass Transfer, 75, p. 381-398.

Kuznetsov, V.V. (2013). Correlation of the Flow Pattern and Flow Boiling Heat Transfer in Microchannels. Heat Transf. Eng., 34(2-3), pp. 235-245.

Kuszewski, M., Zerby, M. (2012). Next Generation Navy Thermal Management Program. CARDIVNSWC-TR-82-(2002)/12.

Lazarek, G.M., Black, S.H. (1982). Evaporative Heat Transfer, Pressure Drop and Critical Heat Flux in a Small Vertical Tube with R-113. Int. J. Heat Mass Transf., 25(7), pp. 945-960.

Lee, J., Mudawar, I. (2008). Fluid Flow and Heat Transfer Characteristics of Low Temperature Two- Phase Microchannel Heat Sink-Part I: Experimental Methods and Flow Visualization Results. International Journal of Heat and Mass Transfer, 51, 4315-4326.

Lee, J., Mudawar, I. (2009). Low-Temperature Two-Phase Microchannel Cooling for High- heatFlux Thermal Management of Defense Electronics. IEEE Transactions on Components and Packaging Technologies June, 2.

Liang, Q.X. Wang and Narain, A. (2004). Effects of Gravity, Shear and Surface Tension in Internal Condensing Flows: Results from Direct Computational Simulations. Journal of Heat Transfer, 126(5), p. 676-686.

Liu, Z., Winterton, R.H.S. (1991). A General Correlation for Saturated and Subcooled Flow Boiling in Tubes and Annuli, Based on a Nucleate Pool Boiling Equation. Int. J. Heat Mass Transf., 34(11), pp. 2759-2766.

Marcinichen, J.B., and Thome, J.R. (2010). New Novel Green Computer Two-Phase Cooling Cycle: A Model for Its Steady-State Simulation. Proc. 23rd Int. Conf. Effic. Cost, Optim. Simulation, Environ. Impact Energy Syst. ECOS, 3, January.

Mersen (2017, October.), R-Tools [Online]. Avaible:http://epus.mersen.com/solutions/cooling-of-power-electronics/r-tools2/

Mehendale, S.S., Jacobi, M.A., and Shah, R.K. (2000). Fluid Flow and Heat Transfer at Micro- and Meso-Scales with Application to Heat Exchanger Design. Appl. Mech. Rev., 53(7), pp. 175-193.

Moore, B.R. (1993). Ideas from Future Technologies Workshop. Held by ARL/TARDEC, ARLSR

Mishima, K., Hibiki, T. (1996). Some Characteristics of Air-Water Two-Phase Flow in Small Diameter Vertical Tubes. Int. J. Multiph. Flow, 22(4), pp. 703-712.

Mudawar, I., Bharathan, D., Kelly, K., Narumanchi, S. (2009). Two-Phase Spray Cooling of Hybrid Vehicle Electronics. IEEE Transactions on Components and Packaging Technologies, June, 32 (2).

Mudawar, I. (2001). Assessment of High-Heat-Flux Thermal Management Schemes. IEEE Transactions on Components and Packaging Technologies, June, 24 (2).

Mudawar, I. (2001). Assessment of High-Heat-Flux Thermal Management Schemes. Components Packag. Technol. IEEE Trans., 24(2), pp. 122-141.

Naik, R., Mitra, S., and Narain, A. (2015). Steady and Unsteady Simulations that Elucidate Flow Physics and Instability Mechanisms for Annular/Stratified Internal Condensing Flows inside a Channel. Journal of Computational Physics,

Naik, R., Mitra, S., and Narain, A. (2014). Steady and Unsteady Computational Simulations for Annular Internal Condensing Flows in a Channel, in Proceedings of 2014 ASME International Mechanical Engineering Congress and Exposition: Montreal, Canada.

Narain, A., et al. (2004). Direct Computational Simulations for Internal Condensing Flows and Results on Attainability/Stability of Steady Solutions, Their Intrinsic Waviness, And Their Noise Sensitivity. Journal of Applied Mechanics, 71(1), p. 69-88.

Ömeroğlu, G . (2018). Investigation In Electrical And Thermal Efficiency Of An Active Cooling Photovoltaic Thermal (Pv/T) Solar System With Taguchi Method. Bilge International Journal of Science and Technology Research , 2 (1) , 47-55 . DOI: 10.30516/bilgesci.406359

Pan, Z., Weibel, J.A., Garimella, S.V. (2015). A Cost-Effective Modeling Approach for Simulating Phase Change and Flow Boiling in Microchannels. Proc. of ASME 2015 Int'l Technical Conf. and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems, San Francisco, CA, pp. 1-9.

Park, C., Zuo, J. (2004). Hybrid Loop Thermal Bus Technology for Vehicle Thermal Management. Advanced Cooling Technologies Inc., Lancaster, PA.

Park, C., Vallury, A. (2006). Advanced Hybrid Cooling Loop Technology for High Performance Thermal Management. 4th International Energy Conversion Engineering Conference, San Diego, California, 26-29.

Park, C., Jaura, A.K. Thermal Analysis of Cooling System in Hybrid Electric Vehicles. SAE Transactions, SAE-2002-01-0710.

Phillips, R.J. (1990). Microchannel Heat Sinks; In: A Bar-Cohen and A. D. Krous, Editors, Advances in Thermal Modeling of Electronic Components and Systems, Vol.2, ASME, New York.

Pereira A., et al. (2017). Comparison Between Numerical and Analytical Methods of AC Resistance Evaluation for Medium-Frequency Transformers: Validation on a Prototype and Thermal Impact Analysis. Canadian Journal of Electrical and Computer Engineering, vol. 40, no.2, pp. 101-109.

Ponnappan, R., Donovan, B., Chow, L. (2002). High power thermal management issues in spacebased systems. Space Technology and Applications International Forum-STAIF, Albuquerque, New Mexico, February 3-6.

Ross, P.J. (1989). Taguchi Techniques for Quality Engineering, McGraw-Hill, Singapure.

Sepahyar, S. (2019). Influence of Micro-Nucleate Boiling On Annular Flow Regime Heat Transfer Coefficient Values and Flow Parameters–For High Heat-Flux Flow Boiling of Water, PhD thesis, Michigan Technological University.

Sullivan, P.F., Ramadhyani, S., Incropera, F.P. (1992). Extended Surfaces to Enhance Impingement Cooling with Single Circular Liquid Jets. In Proceedings of ASME/JSME Joint Conference on Electronic Packages, 207-215.

Thome, J.R. (2006). State-of-the-Art Overview of Boiling and Two-Phase Flows in

Microchannels. Heat Transf. Eng., 27(9), pp. 4–19.

Thome, J.R., Bar-Cohen, A., Revellin, R., and Zun, I. (2013). Unified Mechanistic Multiscale Mapping of Two-Phase Flow Patterns in Microchannels. Exp. Therm. Fluid Sci., 44, pp. 1-22.

Tran, T.N., Wambsganss, M.W., France, D.M. (1996). Small Circular and Rectangular Channel Boiling with Two Refrigerants. Int. J. Multiph. Flow, 22, pp. 485-498.

Triplett, K.A., et al. (1999). Gas–liquid Two-Phase Flow in Microchannels Part I: Two-Phase Flow Patterns. Int. J. Multiph. Flow, 25(3), pp. 377-394.

Tuckerman, D.B., Pease, R.F.W. (1981). High Performance Heat Sinking for VLSI. IEEE Electron Device Letters, May, 2 (5).

Urciuoli, D., Tipton, C.W., Porschet, D. (2012). Development of a 90 kW, Two-Phase, BiDirectional DC-DC Converter for Power Dense Applications. U.S. Army Research Laboratory, #ADA433112, Adelphi, MD

Willingham, T.C., Mudawar, I. (1992). Forced-Convection Boiling and Critical Heat Flux from a Linear Array of Discrete Heat Sources. Int. J. Heat Mass Transf., 35(11), pp. 2879-2890.