Numerical study of Jet Impingement Subcooled Boiling on the Superheated Surfaces

Kazem Esmailpour\textsuperscript{a*}, Arad Azizi\textsuperscript{b}, Seyed Mostafa Hosseinalipour\textsuperscript{c}

\textit{a. Department of Mechanical Engineering, Islamic Azad University, Damavand Branch, Damavand, Iran. Esmailpour@damavandiau.ac.ir, Tel: 02177799188, Cell:+989120630924}

\textit{b. Department of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran. Aazizi1@binghamton.edu, Tel:+16077688239}

\textit{c. Department of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran, Alipour@iust.ac.ir, Tel: +982177240191, Cell: +989121113155}

*Corresponding Author:

Dr. Kazem Esmailpour, Email: Esmailpour@damavandiau.ac.ir

Research Center of Energy, Water and Environment, Mechanical Engineering Department, Iran University of Science and Technology, Narmak, Tehran, Iran. Zip Code: 16846-13114, Mail Box: 16765-163, Phone: +982177240540-50 (Ex: 2929), Fax: +982177240488.
Abstract

Cooling techniques of superheated surfaces by jet impingement with taking advantage of phase change phenomena i.e. boiling heat transfer has proven to be an efficient method because of its high rates of heat transfer. Furthermore, at a specified heat transfer coefficient, flow required for cooling purposes can reduce two orders of magnitude comparing to free-wall parallel flow which is important regarding to energy and water sustainability issues in various industries. This research mainly concerns numerical simulation of hydrodynamics and heat transfer phenomena regarding phase-change jet impingement on nucleate boiling region. Rensselaer Polytechnic Institute wall boiling model based on Eulerian multiphase model and RNG K-ε turbulence model were employed. Each interfacial term was considered and selected based on proximity to real physical phenomena. The selected model in this research was validated by a previously done confined jet impingement subcooled boiling experiment (dielectric fluid-PF5060). Minimum error of 4% and maximum error of 15% were reached at stagnation point. As parametric study, the effect of jet Reynolds number based on nozzle hydraulic diameter at Re 2500 to 10000 and the effect of standoff distance of jet nozzle from target surface at H/D 2, 4 and 6 were investigated.

Keywords: Jet Impingement, Subcooled Boiling, Eulerian Multiphase Model, Mass Transfer Enhancement, Interfacial terms

1. INTRODUCTION

Jet impingement heat transfer with phase change is classified as one of the most efficient cooling methods for thermal management in various industries. There are number of applications for jet impingement boiling heat transfer such as heat treatment of materials and smart surfaces, cooling of electronic modules, emergency cooling safety systems to name a few [1]. The four main regimes in the jet impingement boiling curve are forced convection (single phase heat transfer), nucleate boiling, transition boiling (shoulder of flux) and film boiling. This study is mostly focused on nucleate boiling since vast majority of the mentioned applications operate on this region [2]. Transient conduction by the displacement of hot liquid bulk in to thermal boundary layer through cyclic bubble nucleation, growth and departure in addition to vapor embryo formation and enhanced convection are the primary heat transfer mechanisms in the nucleate boiling process [3]. Turbulence characteristics which happen due to local jet hydrodynamics will be influenced by the vaporization process as well [4-5]. Karwa et.al. did experimental studies on water jet impingement of hot steel plates. Mechanistic or subscale modeling was implemented by multiphase flow visualization with high speed imaging and fast-response thermocouples were embedded within the surface of stainless steel AISI-type 314 cylinder with the homogeneous initial temperature of 900 degrees Celsius. As bubble dynamics and nucleation sites play a crucial role on jet boiling heat transfer mechanism, the observations confirm that removal of bubbles by
forced convection penetrated by jet flow in the early stages of their growth and rapid condensation of the bubble cells after ebullition form the hot surface avoid the buildup of vapor near the wall which is an essential remark in further analysis [6]. Analytical force balance approach employed by Klausner et. al. demonstrated increase in shear and drag and consequently lift force as a result of increase in flow velocity, degree of subcooling and also heat flux has considerable effect on decrease in bubble size, lifetime and departure diameter. In the jet impingement boiling situation, bubble departure diameter is predicted in O(10) μm and lifetime of O(0.1) ms. It is stated in various studies that current state of the art in high-speed imaging could not be able to fully observe such small scale and short-lived phenomena, and that is why the numerical approach is also used by many researchers in order to reach higher accuracy in understanding of the physics of phase-change jet impingement cooling [7-8]. In steady-state measurements, particular boiling condition can be sustained long enough in order to capture necessary details of bubble dynamics. Nevertheless, high thermal capacity of the superheated surface is a key function in slowing down time-dependent temperature variations regarding to boiling phenomena in various regions. Plus, constant surface temperature is preferable compared to constant heat flux rate since the latter provides further instabilities in the nucleation process [9-10]. Many of the models developed for various parameters in boiling such as interphase interaction terms are based on pool or parallel flow boiling configurations aimed for nuclear engineering applications and only few limited investigations were done on subscale modeling of jet impingement boiling. Omar et.al. developed a mechanistic model in order to predict bubble growth and degeneration i.e. bubble departure diameter considering jet velocity, surface superheat and the degree of subcooling by semi-empirical approach using non-linear regression analysis which decreases inaccuracies caused by applying flow boiling formulations in the jet configuration. However, they emphasized that as these models are based on experimental data over specific range of those parameters, they cannot be used at higher or lower ranges with the same confidence [11-12]. Shin et.al. experimentally investigated single and two phase confined impingement jets on various Reynolds numbers and standoff distances. Dielectric fluid PF-5060 with constant degree of subcooling 25K was used in this experiment. The target surface was Inconel-600 plate with 467 μm thickness. Thermal boundary condition of the plate is based on electrical resistive heating which is monitored by 7 K-type thermocouples for temperature measurements [13]. This experiment is employed later in this paper in order to validate the proposed numerical simulation.

Narumanchi et.al. simulated the nucleate boiling process by jet impingement with the use of numerical methods by ANSYS FLUENT in previous studies. Rensselaer Polytechnic Institute (RPI) Eulerian multiphase model was utilized in order to predict the dispersed flow condition resulted by boiling on the superheated surface and various models were employed in order to simulate interphase interaction terms such as interfacial area concentration, lift, turbulent dispersion force, bubble departure diameter etc. Based on this numerical study, 20% deviation from experiment at stagnation point is considered to be an appropriate error in this configuration due to the complexity of
the involving physics [14]. Abishek et.al. recently did numerical research on jet impingement boiling based on RPI wall boiling model. According to their studies, RNG K-ε model with enhanced wall treatment offers suitable accuracy compared to other RANS based turbulence models. Furthermore, Unal’s formulation by heat transfer controlled model for bubble departure diameter combined with Podowski’s or Cole’s model for bubble departure frequency had the highest precision on their simulation [15].

Shademan et.al recently have investigated flow, heat and mass transfer of a jet issuing from a circular nozzle numerically [16]. They used an Eulerian-Eulerian two-phase flow model to study of boiling phenomenon in impinging jet configuration. Comparison of their results with the experimental results confirmed the validity of their developed model for impinging jets. Qiu et.al have conducted a comprehensive review on jet impinging boiling [17]. Their article mainly focuses on experimental studies, but the RPI model is also discussed briefly. Choo et.al. [18] investigated a three dimensional conjugation heat transfer problem involving water boiling process experimentally and numerically on the hot tube array configuration. They simulated water boiling by RPI model. They mainly focused on the empirical study and effect of air and water flow rate was studied as key parameters on the flow and thermal characteristics.

According to our literature review, further analysis in the numerical simulation of jet impingement subcooled boiling and diverse parametric studies can be a key element on better understanding of the underlying physics of this configuration as only few researches have been done in this area. As the effects of jet Reynolds number and standoff distance from target surface are two of the most important parameters in jet impingement research, these criteria were investigated in this paper. In order to measure the precision of the results from the numerical framework, a case study based on experimental data were carried out.

2. GEOMETRY AND COMPUTATIONAL DOMAIN

The geometry in this study was based on confined jet with slot nozzle. As the computational domain was symmetric, only half of the geometry is considered for the simulations to decrease computational expenses. Gravity was taken into account since it is an important parameter regarding the involving physics. No-slip condition is applied on the walls. Fluid characteristics varying with temperature and density gradients are modeled based on Boussinesq approximation. Thermophysical properties of the working fluid which used in this study are available at Table 1.

Figure 1 displays the configuration of the parametric studies. Water is used as working fluid with 20 degrees Kelvin of subcooling. Nozzle diameter is 2mm. Target surface consists of two parts. First part is a copper plate with width of 6mm and thickness of 200μm and depth of 10mm and Second part is two polycarbonate plates with the same size along the surface for insulation.

As mentioned earlier, the computational domain is also non-uniform in this meshing also in order to reach higher accuracy combined with minimum computational expense.
Mesh independency tests based on parameters with higher sensitivity such as vapor volume fraction were performed in each case in order to reach optimum amount of cells. 100×690 cells were approximately used for H/D=4 configuration.

$y^+$ of the first cell of the liquid phase in the stagnation point was approximately 0.2 and it reached maximum of 1.6 along the plate. $y^+$ of the vapor phase based on the chosen thermal boundary condition was less than 0.5.

3. MATHEMATICAL FORMULATION AND MODELING

In this study, subcooled boiling is modeled by the Euler-Euler approach using finite volume method code which has been developed in OpenFoam v4.1. In the Euler-Euler approach, different phases are handled as interpenetrating continua. Phasic volume fraction is introduced in this context which is considered as continues functions of time and space. Wall boiling models such as Rensselar Polytechnic Institute are developed based on Eulerian model which have the most complex formulations in this category. The Eulerian model solves “n” number of conservation equations of mass, momentum and energy for each phase. These governing equations are coupled via pressure and interphase interaction coefficients. These coefficients are treated according to type of phases involved i.e. liquid-gas dispersed flow in this study due to boiling. The interfacial terms could be nonlinear; hence convergence of these models can be slow. Furthermore, the accuracy of the model directly is affected by the closeness of these coefficients to the real physical phenomenon [19].

Kurul and Podowski established the RPI wall boiling model [20]. According to their explanation and figure 3, total wall heat flux consists of three components which are convective heat flux, quenching heat flux and evaporative heat flux with the following formulations.

$$q_w = q_C + q_Q + q_E$$

Convective heat flux express as follows:

$$q_C = h_c (T_w - T_l)(1 - A_b)$$

In the above equation, $h_c$ indicates single phase convective heat transfer coefficient and $T_w$ and $T_l$ are wall and liquid temperatures. The value of $T_l$ is calculated based on fixed $y^+$ of 250 proposed by Egorov and Menter [21]. Quenching heat flux refers to average heat transfer due to instant periodic displacement of cold liquid after detachment of bubble from the surface and it is calculated by equation (3).
\[ q_o = 2K_i(T_w - T_i) / (\pi \lambda_i T)^{0.5} \] (3)

\[ \lambda_i = \frac{K_i}{\rho_i C_{pi}} \] (4)

Where \( K_i, \lambda_i, T \) are thermal conductivity, diffusivity and periodic time (cyclic averaged).

Evaporative heat flux represents by equation (5).

\[ q_e = V_d N_w \rho_v f h_{fe} \] (5)

Where \( V_d \) volume of bubble is based on bubble departure diameter, \( N_w \) is nucleation site density, \( \rho_v \) is vapor density, \( h_{fe} \) is latent heat for vaporization and \( f \) is bubble departure frequency.

Effective area or area of influence is the area occupied by bubbles across nucleation site. \( K \) is an empirical constant which has been modified by Del Valle and Kenning based on subcooled Jacob number [22].

\[ A_p = \min \left( 1, K \frac{N_w \pi D_w^2}{4} \right) \] (6)

\[ K = 4.8 e^{-0.0125 J a_{sub}} \] (7)

\[ J a_{sub} = \rho_l C_{pl} \Delta T_{sub} / \rho_v h_{fe} \] (8)

Where \( \Delta T_{sub} \) is the difference between saturation temperature and liquid temperature.

Bubble departure frequency is calculated by Cole’s photographic study for the pool boiling of distilled water in the near critical heat flux region [23].

\[ f = \frac{1}{T} = \left[ 4g (\rho_l - \rho_v) / 3 \rho_l D_w \right]^{0.5} \] (9)

The effects of wall heat flux, liquid subcooling, liquid velocity and boiling cavity radius on total bubble departure time have been investigated based on dwell time and growth time by Podowski et.al. [24]. This parameter is commonly presented by a semi-empirical
formulation based on wall superheat. Nucleation site density is greatly dependent on surface roughness on microscale. It has almost no effect on liquid temperature, slight influence on gas volume fraction and dominant effect on wall superheat [25].

\[ N_w = C^n (T_w - T_{sat})^n \]  

(10)

Lemmert and Chawla’s empirical coefficients where \( n=1.805 \) and \( c=210 \) which used in this study [26].

There are a number of formulations available for bubble departure diameter. Unal’s semi-empirical correlations based on mechanistic heat transfer controlled bubble model in the “forced-convection surface-boiling” or “partial nucleate boiling” regime seem to have more reliability in predictions compared to other bubble size detachment correlations such as observations by Tolubinsky and Kostanchuk [27,28].

\[ D_w = 2.4210^{-5} \rho^{0.709} \left( \frac{a}{b \sqrt{\varphi}} \right) \]  

(11)

\[ a = \frac{\Delta T_{sup}}{2 \rho_g H_b} \sqrt{\frac{\rho_l C_{ps} K_s}{\pi}} \]

\[ b = \begin{cases} \frac{\Delta T_{sub}}{2 \left(1 - \frac{\rho_g}{\rho_l}\right)} e^{\left(\frac{\Delta T_{sat}}{3}\right)} & \Delta T_{sub} \leq 3 \\ \frac{\Delta T_{sub}}{2 \left(1 - \frac{\rho_g}{\rho_l}\right)} & \Delta T_{sub} > 3 \end{cases} \]

\[ \varphi = \max \left( \frac{U_b}{U_0} \right)^{0.47}, 1.0 \]

Equations 11 refers to Unal’s formulation where \( U_b \) is near wall velocity and \( \Delta T_{sup} \) is wall superheat.

This parameter is different from bubble departure diameter and it refers to average bubble diameter in bulk flow. A correlation by Kurul and Podowski was implemented by user defined function in the simulation [29].
\[ d_b = \begin{cases} 
    d_b^{\text{max}} + \theta \times \Delta T_{\text{sub}} & \text{if } 0 \leq \Delta T_{\text{sub}} \leq \Delta T_{\text{sub}}^{\text{max}} \\
    \max \left( d_b^{\text{min}} \times \exp \left( \frac{\theta}{d_b^{\text{min}}} \Delta T_{\text{sub}}^{\text{max}} \right), 10^{-3} \right) & \text{if } \Delta T_{\text{sub}} \geq \Delta T_{\text{sub}}^{\text{max}} \\
    10^{-3} & \text{if } \Delta T_{\text{sub}} \leq 0 
\end{cases} \] (12)

Where \( d_b^{\text{min}} = 1.5 \times 10^{-4} \, \text{m} \) and \( d_b^{\text{max}} = 10^{-3} \, \text{m} \) are constants in Equation (12).

In the Eulerian multiphase formulation, each phase’s field and constitutive equations are solved separately. However, since averaged fields of each phase are coupled to each other to some degree, the interaction terms appear on the equations with taking into account dynamic and non-equilibrium states. Interfacial momentum, mass and energy transfer are modeled regarding to interfacial transfer conditions and constitutive laws for interactions between phases.

There is a strong relation between interfacial transport terms of mass, momentum and energy to interfacial area concentration. The interfacial area concentration defined as interfacial area between two phases per unit mixture volume which is related to the structure of two phase flow. Algebric formulation by Ishii et.al. for boiling flows was used [30].

\[ A = \frac{6(1-\alpha_p) \min(\alpha_p, \alpha_{\text{pcrit}})}{D_w(1-\min(\alpha_p, \alpha_{\text{pcrit}}))} \] (13)

The interfacial heat transfer coefficient is calculated according to Ranz and Marshall correlation for Nusselt number as follows [31].

\[ Nu_q = 2 + 0.6 \frac{Re_p^{1/2}}{Pr_q^{1/3}} \] (14)

### 3.1. Momentum transfer modeling

Ishii model is employed for Drag coefficient \( C_D \) in bubbly flows which is determined based on the minimum value between viscous region and distorted region [32].

\[ C_D = \min \left( \frac{24}{Re} \left( 1 + 0.15 Re^{0.75} \right), \frac{2}{3} \left( \frac{D_w}{\sqrt[\frac{1}{3}]{\frac{\sigma}{g \left( \rho_p - \rho_q \right)}}} \right) \right) \] (15)
The lift force as a result of interaction of generated bubbles with shear layer is modeled by Moraga et al. formulation. The lift coefficient in their model considers 1. aerodynamic lift resulting from interaction between bubbles and continues phase 2. vorticity induced lift due to interaction between dispersed phase particles and vortices by bubble wake [33].

\[
C_l = \begin{cases} 
0.0767 & \phi \leq 6000 \\
0.12 - 0.2e^{-\frac{\phi}{3.6 \times 10^7}} & 6000 < \phi < 5 \times 10^7 \\
-0.6353 & \phi \geq 5 \times 10^7 
\end{cases}
\]  

(16)

Where \( \phi \) is the product of bubble Reynolds number, \( Re_b \), and vorticity Reynolds number, \( Re_v \)

Turbulent dispersion force is as a result of chaotic liquid velocity fluctuations and acts as turbulent diffusion in turbulent dispersed flows. Burns et al. came up with a formulation from favre averaging of the interfacial drag force [34]. The modified two phase dispersed flow formulation is shown below.

\[
F_{q, disp} = -C_{q_p} \frac{v_{q_p}}{\sigma_{TD}} \left( \frac{1}{\alpha_q} + \frac{1}{\alpha_p} \right) \nabla \sigma_q
\]  

(17)

\( C_{q_p} \) is interphase exchange coefficient for the interfacial drag force, \( v_{q_p} \) is kinematic eddy viscosity of the dispersed phase and \( \sigma_{TD} \) is turbulent Prandtl number for volume fraction dispersion.

3.2. Modeling of turbulence

Prediction of turbulent flow in the impinging jets has some difficulties due to complex flow hydrodynamics in this configuration. Jet expansion angle, interaction between inlet flow from nozzle and static fluid in the surrounding, turbulence intensity increase in the direction of the jet axis, possible relaminarization around stagnation region, possible transition from laminar to turbulent flow in the wall jet region are just some examples of the mentioned complexity [35].

Several investigations took place by researchers in the past decade on accurate modeling of heat transfer and fluid flow in the impinging jets [36]. RNG K-\( \varepsilon \) RANS based turbulence model combined with enhanced wall treatment was employed in this research due to high accuracy of predictions in addition to computational expense. Necessary precautions for mesh size regarding \( y+ \) limit based on the chosen wall treatment was performed.

The influence of dispersed phase on the turbulent flow can is added by two source terms
in the modified $k-\varepsilon$ equations modeled by Troshko Hassan [37].

$$\pi_{K_n} = C_{kr} \sum_{p=1}^{M} K_{eq} \left[ \overline{U}_p - \overline{U}_q \right]^2$$  \hspace{1cm} (18)

$$\pi_{\varepsilon_n} = C_{\varepsilon} \frac{1}{\tau_p} \pi_{K_n}$$  \hspace{1cm} (19)

4. Solution method

Pressure and velocity coupling were resulted from coupled algorithm in this simulation. Discretization of momentum, energy, turbulent kinetic energy and turbulent dissipation rate were achieved from Quadratic Upstream Interpolation for Convective Kinematics (QUICK) scheme. In addition, modified High Resolution Interface Capturing (HRIC) was employed for volume fraction discretization. Convergence criterion for all parameters was defined as Eq. (20):

$$\left| \frac{\varphi_{n+1} - \varphi_n}{\varphi_n} \right| \leq 10^{-6}$$  \hspace{1cm} (20)

5. Validation of the numerical framework

The numerical model on this research was validated by experimental study of Shin et.al. [13]. Jet Reynolds number based on hydraulic diameter is 1999 and nozzle standoff distance from target surface is 8mm (H/D=4). The configuration consists of an Inconel plate with depth of 8mm and width of 50mm which the first 10mm are heated by resistive heating. Volumetric heat generation is employed in order to model thermal boundary conditions of this problem. Atmospheric condition is applied for pressure and PF-5060 liquid inlet temperature is 304.15 K. Polycarbonate plates were used as insulation based on the schematic given at figure 4.

No-uniform meshing was employed with higher concentration of computational cells in the areas with higher gradients such as stagnation region. Roughly cells were used in this mesh which is shown on figure 2.
6. RESULT AND DISCUSSION

6.1. Validation of the numerical framework

Figure 6 clarifies the comparison of boiling curves obtained in the present study against experimental boiling result of Shin et al. [13]. The surface averaged of total heat flux was calculated after convergence and then in terms of the saturation temperature difference is plotted.

Minimum and maximum deviations of 4\% and 15\% were reached respectively based on the following error estimation:

\[
\text{Error \%} = \left[1 - \left(\frac{T_{\text{wall \ Numerical}}}{T_{\text{wall \ Experimental}}(\text{Stagnation \ point})}\right)\right] \times 100
\]

(21)

Reasons for the deviation of numerical data from experimental results are discussed in brief in the following paragraphs. Semi-empirical formulation for nucleation site density by Lemmert and Chawla which was used in this research is based on water as working fluid [23]. However, PF-5060 liquid in the validation is a dielectric fluid. As dielectric fluids have significantly higher amounts of wetting, some nucleation sites become deactivated. As superheat increases, these nucleation sites become activated again which is observed as overshoot phenomenon in the boiling curve [38]. Surface tension massively increases as bubbles become smaller. This situation happens in jet impingement boiling due to high turbulence intensities and momentum transport which results in smaller departure diameters comparing too pool boiling. Bubbles with high surface tension maintain their spherical shape. This analysis confirms the model chosen for lift force as Moraga’s formulation is based on spherical shape for the dispersed phase [30]. However, as the surface tension was considered an average in the numerical model, slight error happens as a result of this averaging. Each of the empirical-analytical formulations in the simulation of boiling phenomenon such as bubble departure frequency and diameter are valid for certain range of wall superheat, Reynolds number etc. and they cannot be used with the same confidence on various conditions. As it is not possible to obtain an individual correlation for each parameter for each test run, hence slight error to some degree is inevitable. In addition to errors regarding two-phase flow, turbulence modeling by Reynolds-averaged Navier–Stokes (RANS) equations and Boussinesq approximation results in inaccuracies. As it has been stated in many researches, accurate prediction of turbulent flow and heat transfer has noticeable amount of importance in achieving reliable results. It should be noted that Direct Numerical Simulation (DNS) of this problem is still not affordable by today’s computational capabilities as hundreds of particles should be analyzed individually [39].

Minimum secondary phase (vapor) thickness of 60 \( \mu \)m at stagnation region and maximum thickness of 80 \( \mu \)m at wall region were observed as shown in figure 7. This
analysis on vapor volume fraction along the wall is only available on numerical studies which complement the experimental research.

**6.2. Effect of Jet Reynolds number**

The effect of jet Reynolds number based on hydraulic diameter on the rate of heat removal from target surface and the thermo-hydrodynamics of two-phase flow was studied. Fully developed velocity profile was employed on the nozzle outlet. Three Reynolds numbers of 2500, 5000 and 10000 were investigated in this configuration. Constant temperature boundary condition and fixed standoff distance of 8mm (H/D=4) from the plate was used. The range of superheat is from -5 to 16.5 K. The inlet temperature of water is approximately 353 K with zero volume fraction of vapor. Comparison of the boiling curves based on area weighted average of total heat flux can be seen on figure 8.

According to the extracted results, rate of heat transfer between heated plate and the impinging jet was improved noticeably as jet velocity increased. Jet Reynolds of 10000 has the highest rate of cooling versus jet Reynolds of 2500 which has the lowest rate as predicted. This conclusion can also be seen on figure 9 as total heat flux of various Reynolds numbers are shown along the first 10 mm of surface from the stagnation point.

As shown in figure 9 at the center of the impinging jet, the stagnation point has the highest rate of cooling (maximum heat removal), and by getting away from position zero, the rate of heat transfer decreases along x-direction. By reaching the third millimeter, the junction plate is polycarbonate with almost zero heat transfer.

Temperature and volume fraction distributions for different Reynolds numbers are displayed in figure 10. Based on the temperature curve, the temperature of the copper surface on three Reynolds are approximately equal with constant temperature boundary condition. However, Reynolds 2500 shows 1 K higher surface temperature on stagnation point compared to Reynolds 10000 which verifies the stated analysis on jet boiling thermo-hydrodynamics. On the right part of figure 10 by increasing jet velocity it was observed that rate of heat transfer increases. As a result, flow velocity increases on the wall which results in less bubble growth time and less number of bubbles would have the opportunity to collide and form larger bubbles. This forced convection leads to smaller bubble departure diameters and more condensation which results in decrease in vapor volume fraction on the copper plate. This trend changes after the third millimeter due to jet hydrodynamics at polycarbonate junction. The reason for this issue is described by figure 11.

As jet Reynolds increases, the momentum boundary layer becomes thinner; as a result the flow pushes the vapor towards the wall. An extra velocity increase also appears on the wall region due to impingement jet physics. The peak in the fourth millimeter of
vapor volume fraction happens because of this phenomenon.

Secondary phase thickness at various Reynolds numbers is shown at figure 11. As can be seen, Reynolds numbers of 2500 and 10000 represent maximum and minimum values of thickness respectively. Noticeable separation of flow was observed on the wall region according to given streamlines. These are called secondary vortices which are associated with pressure gradient fluctuations based on Lior and Zuckerman explanation [33]. Secondary peak raises local turbulence intensity in the flow which leads to increase in heat and mass transfer rates. However, overall flow kinetic energy decreases on downstream due to this phenomenon.

6.3. Effect of H/D

Ratio of the nozzle distance from target surface to nozzle hydraulic diameter is considered an important parameter in jet impingement studies. The effect of standoff distance was investigated at H/D of 2, 4, and 6 in this research. This study conducted at fixed Reynolds number of 5000 and similar to previous parametric study, fully developed velocity profile was employed at nozzle exit. The employed configuration in terms of materials, dimensions of target surface and insulation was the same as jet Reynolds number study. A comparison of the boiling curves based on area weighted average of total heat flux is shown on figure 12.

By reviewing boiling curves regarding to H/D parameter, it can be seen that with decrease in nozzle to target surface distance, the amount of total heat transfer was increased. Thus, H/D=2 has the highest rate of cooling and H/D=6 has the minimum rate. This concept is displayed on figure 13 by total heat flux from stagnation point to 10 mm distance along the plate.

Analogous trend to Reynolds study for surface temperature and vapor volume fraction is observable for various H/D as it’s shown in figure 14.

As it can be seen on the left, as H/D increases, the rate of cooling decreases which resulted in escalation in surface temperature on the copper plate. Temperature difference of 0.7 K was found on stagnation point. On the left, vapor volume fraction along the surface on the smallest H/D is minimum compared to higher H/D ratios which are 4 and 6. This is explained by velocity contours on figure 15.

As H/D ratio declines, length of potential core shortens which results in positive flow acceleration of the wall and eventually higher rate of cooling. Table 2 summarizes secondary phase thickness values at stagnation point and junction (3rd mm) for Reynolds and H/D studies.

7. Conclusion
A solution was proposed in order to simulate turbulent jet impingement subcooled boiling flow in the confined configuration by employing Eulerian multiphase model. RPI wall boiling model and RNG k-ε turbulence model was utilized by considering appropriate interfacial terms according to the problem. Validation of the numerical framework took place by experimental study of Shin et.al. Minimum error of 4% and maximum deviation of 15% was observed at stagnation point. The effect of jet Reynolds number based on hydraulic diameter (Re 2500, 5000 and 10000) and nozzle standoff distance from target surface (H/D 2, 4 and 6) were discussed and analyzed in detail. Various existing and novel graphs and contours were provided in order to deepen current perception of this issue. Our studies indicate that by increasing jet Reynolds number and decreasing standoff distance, total rate of cooling increases on the target surface and vice versa. Total heat flux curves coupled with thermo-hydrodynamic illustrations confirm reported conclusions. As numerical simulation of jet impingement with phase change is only addressed recently by researchers due to high volume of complexity in physics and great deal of involving parameters, further investigation in this area is still necessary. However, current research attempted to investigate main variables and also introduce new methods of reporting results which are key elements in jet impingement subcooled boiling studies.

1. Acknowledgements

This research was supported financially by The Vice-Chancellery for Research and Technology of Islamic Azad University, Damavand branch. We are thankful to our colleagues Professor Mostafa Hosseinalipour and Arad Azizi who provided expertise that greatly assisted the research.

NOMENCLATURE

\[ p \] Continues phase
\[ q \] Dispersed phase
\[ \rho \] Density (Kg/m³)
\[ K \] Thermal conductivity (W/m-K)
\[ c_p \] Specific heat (J/Kg-K)
\[ \mu \] Viscosity (Kg/m.s)
\[ \sigma \] Surface tension (N/m)
\[ L \] Latent heat (J/Kg)
\[ h_c \] Convective HT coefficient (J/Kg)
\[ T \] Temperature (K)
\[ w \] Wall (-)
\[ l \] Liquid phase (-)
\[ v \] Vapor phase (-)
\( T_{sat} \) Saturation temperature \( (K) \)

\( \Delta T_{sup} \) Wall superheat \( (K) \)

\( \lambda \) Diffusivity \( (m^2/s) \)

\( V_d \) Volume of bubble \( (m^3) \)

\( T \) Periodic time \( (s) \)

\( h_{fc} \) Latent heat of vaporization \( (J/Kg) \)

\( N_w \) Nucleation site density \( (m^2) \)

\( f \) Bubble departure frequency \( (s^{-1}) \)

\( \Delta T_{sub} \) Degree of subcooling \( (K) \)

\( U \) Velocity \( (m/s) \)

\( D_w \) Bubble departure diameter \( (m) \)

\( d_b \) Bubble diameter \( (m) \)

\( \alpha \) Volume fraction \( (-) \)

\( A \) Area \( (m^2) \)

\( C_D \) Drag coefficient \( (-) \)

\( C_l \) Lift coefficient \( (-) \)

\( Re \) Reynolds number \( (-) \)

\( C_{qp} \) Interphase exchange coefficient \( (-) \)

\( v_{sq} \) Kinematic eddy viscosity \( (m^2/s) \)

\( \sigma_{TD} \) Turbulent dispersion Prandtl \( (-) \)

\( F_{disp} \) Turbulent dispersion force \( (N/m) \)

\( y^* \) y-plus \( (-) \)

\( q_C \) Convective heat flux \( (J/s) \)

\( q_Q \) Quenching heat flux \( (J/s) \)

\( q_E \) Evaporative heat flux \( (J/s) \)

\( q_T \) Total heat flux \( (J/s) \)

\( Ja \) Jakob number \( (-) \)

REFERENCES


Biographical Information

Arad Azizi is Ph.D. student in the department of mechanical engineering at State University of New York at Binghamton, NY, USA. This paper is based on his master’s thesis research (2014-16) at mechanical engineering department of Iran University of Science and Technology. He has done various researches regarding forced convective cooling, thermal analysis and solar energy conversion systems. His research interests are microscale heat and mass transport, multiphase flows and sustainability.

Kazem Esmailpour is assistant professor of thermo-fluid engineering at Islamic Azad University of Damavand branch, Iran. He obtained a Ph.D. in mechanical engineering from Iran University of Science and Technology (IUST) in 2014. He has carried out researches in single-phase and two-phase impinging and opposing jet contactors, thermal system optimization and heat transfer enhancement. He is a fellow at Research Center of Energy, Water and Environment in IUST, Tehran, Iran. Email: kazem.esmailpour@gmail.com

Seyed Mostafa Hosseinalipour is associate professor in the department of mechanical engineering and head of aerospace group at Iran University of Science and Technology, Tehran, Iran. He is also director of Research Center of Energy, Water and Environment and office of sustainability at IUST. He obtained his Ph.D. from department of mechanical engineering at McGill University, Montreal, Canada (1996). His main research areas are optimization and design of energy consuming and energy conversion systems, CFD and parallel processing, impinging jets and drying. Email: alipour@iust.ac.ir
Figures and tables captions

Table 1 Thermo-physical properties of the working fluid

Figure 1 Schematic of the configuration considered for parametric studies

Figure 2 Meshing and computational domain of the configuration considered for parametric study H/D=4.

Figure 3 Schematic of RPI wall boiling model

Figure 4 Schematic of the configuration considered for validation.

Figure 5 Meshing and computational domain of the configuration considered for validation

Figure 6 Validation of numerical simulation by Shin et.al. experimental study[13]

Figure 7 a) Temperature contour with streamlines, b) vapor volume fraction contours with streamlines at stagnation region and c) wall region of the numerical validation at stagnation point temperature of approximately 341 K

Figure 8 Total heat flux for various jet Reynolds number based on nozzle hydraulic diameter

Figure 9 Total heat flux (W/cm2) from the stagnation point to the first 10 mm along the surface at temperature of approximately 387 K

Figure 10 a) the surface temperature in Kelvin and b) vapor volume fraction from the stagnation point to the first 10 mm the along plate direction at constant temperature of approximately 387 K

Figure 11 Vapor volume fraction contours at stagnation region (fist row) and wall region (second row) at constant temperature of 387 K at jet Reynolds numbers from left to right: 2500, 5000 and 10000

Figure 12 Total heat flux for various distance of jet nozzle from target surface

Figure 13 Total heat flux (W/cm2) from the stagnation point to the first 10 mm along the surface at temperature of approximately 387 K

Figure 14 a) Surface temperature on the copper plate in Kelvin and b) vapor volume fraction from the stagnation point along plate direction at constant temperature of approximately 387 K

Figure 15 Liquid velocity contours at constant temperature of 387 K at H/D from left to right: 2, 4, and 6

Table 2 Comparison of the dispersed phase (vapor) thickness on the target surface for
<table>
<thead>
<tr>
<th>Property</th>
<th>Units</th>
<th>PF-5060 liquid</th>
<th>PF-5060 vapor</th>
<th>Water liquid</th>
<th>Water vapor</th>
<th>Inconel 600</th>
<th>Copper</th>
<th>Polycarbonate</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$</td>
<td>Kg/m$^3$</td>
<td>1658.836</td>
<td>13.4</td>
<td>965.73</td>
<td>0.5976</td>
<td>8470</td>
<td>8978</td>
<td>1210</td>
</tr>
<tr>
<td>$K$</td>
<td>W/m-K</td>
<td>0.05658</td>
<td>0.02</td>
<td>0.675</td>
<td>0.02512</td>
<td>14.9</td>
<td>387.6</td>
<td>0.22</td>
</tr>
<tr>
<td>$C_p$</td>
<td>J/Kg-K</td>
<td>1062.347</td>
<td>500</td>
<td>4205.54</td>
<td>2078.18</td>
<td>444</td>
<td>381.0</td>
<td>1250</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Kg/m.s</td>
<td>$5.65 \times 10^{-4}$</td>
<td>$1.81 \times 10^{-5}$</td>
<td>$3.156 \times 10^{-4}$</td>
<td>$1.227 \times 10^{-4}$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$T_{sat}$</td>
<td>K</td>
<td>329.15</td>
<td></td>
<td>373.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\sigma$</td>
<td>N/m</td>
<td>0.00827</td>
<td></td>
<td>0.059</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>L</td>
<td>J/kg</td>
<td>97000</td>
<td></td>
<td>2257000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 1

Figure 2

Figure 3

Figure 4
Figure 6

Figure 7

Figure 8
Figure 11

Figure 12
Figure 13

Figure 14
### Table 4

<table>
<thead>
<tr>
<th>Reynolds 2500</th>
<th>Stagnation point</th>
<th>Junction (3rd mm from center)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>96 μm</td>
<td>140 μm</td>
</tr>
<tr>
<td>Reynolds 5000</td>
<td>52 μm</td>
<td>103 μm</td>
</tr>
<tr>
<td>Reynolds 10000</td>
<td>41 μm</td>
<td>72 μm</td>
</tr>
<tr>
<td>H/D 2</td>
<td>48 μm</td>
<td>90 μm</td>
</tr>
<tr>
<td>H/D 4</td>
<td>52 μm</td>
<td>103 μm</td>
</tr>
<tr>
<td>H/D 6</td>
<td>59 μm</td>
<td>105 μm</td>
</tr>
</tbody>
</table>

**Figure 15**

Table 4