The jet impingement boiling heat transfer with ad hoc wall thermal boundary conditions

Qiu, Lu; Dubey, Swapnil; Choo, Fook Hoong; Duan, Fei

2016

Qiu, L., Dubey, S., Choo, F. H., & Duan, F. (2016). The jet impingement boiling heat transfer with ad hoc wall thermal boundary conditions. *Applied Thermal Engineering*, 108, 456-465.

<https://hdl.handle.net/10356/83600>

<https://doi.org/10.1016/j.applthermaleng.2016.07.134>

© 2016 Elsevier Ltd. This is the author created version of a work that has been peer reviewed and accepted for publication by *Applied Thermal Engineering*, Elsevier Ltd. It incorporates referee's comments but changes resulting from the publishing process, such as copyediting, structural formatting, may not be reflected in this document. The published version is available at: [<http://dx.doi.org/10.1016/j.applthermaleng.2016.07.134>].

Downloaded on 26 Apr 2023 21:38:45 SGT

The jet impingement boiling heat transfer with ad hoc wall thermal boundary conditions

Lu Qiu^{a,b}, Swapnil Dubey^a, Fook Hoong Choo^a, Fei Duan^{b,*}

^a*Energy Research Institute @ NTU, Nanyang Technological University, 1 Cleantech Loop, 06-04 Cleantech One, Singapore 637141*

^b*School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798*

Abstract

In a two-phase heat exchanger, the thermal boundary condition at the boiling wall plays an import role. To investigate the characteristics of its flow and heat transfer, it is required to solve a three-domain conjugation heat transfer problem which takes into account of boiling, conduction, and air convection. In the current design, the saturated water flows into a cylindrical chamber with a tube array, whereas the hot air travels outside of chamber and boils the water inside. The effects of the water inlet velocity and hot air inlet mass flow rate are measured in the experiments. A simulation tool with Graphical User Interface code is developed for predicting the three-domain conjugation heat transfer. The boiling heat transfer in the complex case is showed to be well explained in the approach of combining the Rensselaer Polytechnic Institute (RPI) boiling model. The experiments indicate that the wall temperature on the solid-air interface and the transferred energy are independent of water inlet velocity but significantly depend on the air inlet mass flow rate. The wall temperatures in the centre core area (tube array region) are relatively uniform, whereas a huge temperature gradient is measured in the peripheral area. The maximum temperature difference in the core region is only around 17.7% or 24.6% of the core-to-peripheral temperature difference in the cases with the high or low air inlet velocity. The experimental observations have been reproduced in the simulation. The heat transfers on the hot air side and the water boiling side significantly influence each other. Prominent variations of wall temperature and heat flux result in a co-existence of single-phase water convection and the water boiling flow. It demonstrates that the conjugation has to be considered with applying an ad hoc thermal boundary conditions in the cases.

Keywords: Conjugation heat transfer, jet impingement boiling, RPI boiling model

*Tel: +65 6790 5510; Fax: +65 6792 4062
Email address: feiduan@ntu.edu.sg (Fei Duan)

Nomenclature

A	heat transfer surface area	Subscripts	
A_b	nucleate bubble covered area	air	air side parameter
C_p	constant pressure heat capacity	wat	water side parameter
d	diameter	CB	convective heat flux in the boiling model
D_w	bubble departure diameter	$cell$	first cell adjacent to the wall
f	bubble departure frequency	EB	evaporative heat flux in the boiling model
g	gravity	e	environment
h	heat transfer coefficient	exp	experimental results
Ja	Jacob number	L	latent heat
$\dot{m}$	mass flow rate	l	liquid
N_w	Nucleate site density	$loss$	loss
q''	heat flux	f	flow parameter
Q	heat energy	sol	solid domain parameter
Re	Reynolds number	sat	saturation
T	temperature	sim	numerical results
U	velocity	sub	subcooling
X, Y, Z	coordinates	QB	quenching heat flux in the boiling model
y^+	dimensionless wall distance	in	inlet parameter
α	heat loss coefficient	out	outlet parameter
ρ	density	v	vapour
λ	thermal conductivity	w	wall
Φ	vapour volume fraction		

1. Introduction

Nucleate boiling is widely employed in many industrial applications due to its high heat transfer coefficient. The near-wall liquid evaporation absorbs the energy from the wall, whereas the boiling generated bubbly flow enhances the flow mixing [1, 2]. Given that the wall heat flux is sensitive to the wall temperature in the nucleate boiling regime, the conduction inside the hot wall as well as the thermal boundary condition applied to the solid domain are crucial to the boiling heat transfer [3]. The thermograph measurements proved that a significant temperature variation could be generated on the boiling surface [4], indicating that the different flow regimes may be co-existing. For example, the co-existence of single-phase convection and the nucleate boiling in a jet impingement boiling

configuration was observed [4, 5]. Therefore, solving a conjugation heat transfer problem may be necessary in many nucleate boiling applications. For example, in the heat exchanger of a thermal acoustic Stirling engine, a cylindrical heat exchanger filled with the dense hollow circular tubes was employed [6, 7]. The saturated liquid flowed across the space outside of tube array to cool down the gas fluid flow inside the tubes. In this distinguished configuration, investigating the boiling heat transfer alone may be not enough since the hot gas flow on the other side supplied a complex and unique thermal boundary condition for the boiling heat transfer domain. Therefore, a comprehensive conjugation heat transfer problem should be further concerned.

When supplying the liquid into the hot shell-side cylindrical chamber from a circular inlet, a jet impingement boiling configuration is formed. Investigations regarding jet impingement boiling have been attracted more attentions [1, 2]. The boiling heat transfer is influenced by not only the jet parameter but also the surface conditions. The effects of jet velocity [8], jet subcooling [9], jet diameter [10], jet quantity [4], micro-jet [11], cross flow [12], nanofluids [13] and surface conditions [8] in the free surface jet impingement boiling configuration were investigated. A good agreement was reached that increasing the jet velocity and subcooling did not influence the boiling curve but promoted the critical heat flux (CHF). Surface coating enhances the two parameters significantly [14, 15]. Nanofluids jet impingement boiling was also investigated [13], the deposition of the nano-particles on the surface changes the surface condition notably [16]. Contradictive findings were reported for the submerged and confined configurations. For example, the boiling curve was found to be either independent of jet velocity [17–19] or a function of the jet velocity [20–22]; either independent of jet subcooling [18] or a function of the subcooling [20]. The effects of impact distance [19], multiple jets [19], surface conditions [18, 21] and the micro-jet impingement [23] were investigated as well. Numerical simulations regarding jet impingement boiling,

unlike the experimental investigations, were rare. Although modelling boiling heat transfer was complex, attempts were made by adding the artificial diffusivity to the single-phase equations [24, 25], or adding inter-phase mass transfer rate correlations in Eulerian based two-phase equations [26]. Aside from them, the Rensselaer Polytechnic Institute (RPI) boiling model was one perspective mechanism model to numerically solve the jet impingement boiling problems [3, 27, 28]. However, to our best knowledge it has not been used in simulating the boiling heat transfer in complex flow configurations.

The aforementioned boiling regime variation across the surface was more prominent in the jet impingement boiling heat transfer configurations. The jet impingement produced the diverse convective heat transfer across the surface, which in turn rendered a high temperature gradient if an iso-flux boundary condition was imposed. However, the conduction inside the thick wall decreased the surface temperature gradient, thus the conjugation mattered in the scenario [3]. For example, different phenomena could be observed in the experiments if the boiling wall was made with a thin film heater or a heated copper-block. The co-existence of single-phase convection and nucleate boiling was observed in the non-conjugation configurations [4, 5], which could be numerically predicted by the RPI boiling model [3].

To sum up, the conduction inside the solid wall is important to the boiling heat transfer. The thermal boundary condition imposed on the wall may influence the boiling heat transfer as well. Therefore, a comprehensive three-domain conjugation heat transfer problem is investigated in order to show the significance made by the conjugation. Since the simulation platform for this kind of conjugation problems is not commercially available. A simulation tool has been developed based on ANSYS-Fluent V16.1 and Graphical User Interface (GUI) code and validated with the experimental data. Finally, the experimental results are explained by the numerical simulations.

2. Problem description

As shown in Fig.1, a three-domain conjugation heat transfer problem is studied, which includes a two-phase boiling flow zone, a solid conduction zone and a single-phase convection zone. On the air side, the high-temperature air travels through a circular channel ($d=38$ mm), then enters fifty-five small tubes ($d=1.4$ mm), and expands to another circular channel ($d=38$ mm) in sequence. Outside the tubes, the saturated liquid impinges onto the tube array from a circular inlet ($d=6$ mm) and then boils into steam. The cross section of the cylindrical water chamber has a diameter of 44 mm. The front or back side of the boiling chamber has a 3 mm thick solid wall on which the tubes are fixed. The materials of the solid parts are stainless steel. The boiling water domain and the hot gas domain are completely separated by the solid domain. The arrangement of the staggered tubes is shown in Figs.1 (b) and (d). The hot air flows in horizontal direction (Z-direction), whereas the saturated water enters and exits the cavity vertically (X-direction) against the gravity. The physical properties of working fluid and the flow parameters are summarized in Table 1. The water inlet temperature, $T_{in, wat}$, was fixed at 97.5 °C, whereas the water inlet velocity, $U_{in, wat}$, varied from 0.1 m/s to 0.4 m/s. The inlet temperature of air flow, $T_{in, air}$, varied from around 100 °C to around 230 °C, whereas the air inlet mass flow rate, $\dot{m}_{in, air}$, changed from 0.002 to 0.006 kg/s. Since the density of the air changed significantly with temperature, the air inlet mass flow rate rather than the velocity was controlled. The maximum inlet air velocity was 7.5 m/s, however, it was accelerated to 100.7 m/s in the small tubes. The related water and air Reynolds numbers are calculated and listed in Table 1 as well.

3. Materials

3.1. Experiment facility and procedure

Fig.2 illustrates the schematic of the experimental system. The air was heated with an air heater and then supplied into a 110 mm long insulated straight channel with an inner diameter of 38 mm. A mesh was assembled at the entrance of the straight channel. The hot air hit the stainless steel heat exchanger and then passed through the small holes. A straight channel (50 mm in length) and a converging channel (30 mm in length) were connected after the heat exchanger. The channel before and after the heat exchanger were made by 8 mm thick Teflon that covered by a 10 mm thick insulation. Since the temperature before the heat exchanger was relatively uniform, one K-type thermocouple was installed in the centre of the channel that 25 mm in the front of the heat exchanger to measure the inlet air temperature, $T_{in,air}$. After the heat exchanger, a significant span-wise temperature difference might exist. Therefore, a flow converging section was added at the outlet in order to measure the mean outlet temperature, $T_{out,air}$. The mass flow rate was measured with a calibrated rotameter before the air heater, being double-checking with the calculation on the basis of enthalpy increase before and after the heater. The deionized water was heated up to 99.5 °C in the reservoir and then pumped into the test section. Although the tubes were well insulated, heat loss was inevitable. The measured inlet water temperature, $T_{in,water}$, was around 97.5 °C. Inside of the chamber, the water passed through the hot tubes and boiled into vapour. The exhausted two-phase flow was condensed back to liquid before the flow meter. At the end, the water was circulated back to the reservoir for the next cycle. As illustrated in Fig.2, four K-type thermocouples were welded on the external surface of the heat exchanger (solid-air interface). Three of them, $T_{w,1}$, $T_{w,2}$ and $T_{w,3}$, were close to the tube array, whereas the last one, $T_{w,4}$, was close to the outlet.

The enthalpy change from the inlet to the outlet of the heat exchanger could be divided into two parts: the net heat energy that transferred from the air to water, Q_w , and the heat loss, Q_{loss} . Therefore, the net heat energy can be calculated as follows,

$$Q_w = C_p \dot{m}_{air} (T_{in,air} - T_{out,air}) - Q_{loss} \quad (1)$$

where the C_p is the heat capacity that evaluated with the average air temperature of inlet and outlet, and $\dot{m}_{air}$ is the mass flow rate of the air. The heat loss was estimated from the temperature difference between the mean air temperature, T_f , and the ambient temperature, T_e , by following,

$$Q_{loss} = \alpha_{loss} (T_f - T_e) \quad (2)$$

where α_{loss} is the heat loss coefficient. It was calibrated in a separated heat loss experiment in which the hot air was passing through the 38 mm channel without the heat exchanger. In this case, the temperature difference between inlet and outlet was purely induced by the heat loss. During the experiments, the heating power of the air heater was adjusted to increase the air inlet temperature from around 100 °C to 230 °C. After that, the channel was cooled down to the initial state, and then we repeated the experiments at another fluid inlet condition. It was calibrated that the errors of all the temperature measurements were less than ± 0.5 °C. The uncertainties of the mass flow rate of water and air were 2.0% and 1.5%, respectively. The absolute and relative uncertainties of the net transferred energy Q_w were 3.2 W and 2.2% in the case of the highest air inlet temperature.

3.2. Numerical simulations

3.2.1. RPI boiling model

In the three-domain conjugation heat transfer problem, the air-side single-phase convection and the solid domain conduction are relatively simple to model, and the conventional standard Navier-Stokes (N-S) equation solver is able to make the simulation of the flow and heat transfer in good precision. A standard K-omega turbulence model is employed here. The boiling heat transfer, however, is complicated to be modelled. As mentioned, only a few of works would be found in the literatures. In the current investigation, the RPI boiling model that embedded in ANSYS Fluent V16.1 is used to simulate the boiling flow and heat transfer on the water side [29]. It operates based on an Eulerian multi-phase framework in which both the liquid and vapour phases are treated as continues flows. Two sets of N-S equations are solved separately, but the sum of the volume fraction of the two phases equals to 1.0 in every control volume. Boiling induced momentum and energy exchanges between two phases are added as the source terms in the N-S equations. In the same way, the bubble induced turbulence is included in K-epsilon equations as the source terms. The mass transfer between the two phases is calculated with the evaporation-condensation model. The RPI boiling model is a kind of wall heat flux partitioning model in which the wall heat flux is separated into three parts: single-phase convective heat flux, q''_{CB} , evaporative heat flux, q''_{EB} , and quenching heat flux, q''_{QB} .

$$q''_w = q''_{CB} + q''_{QB} + q''_{EB} \quad (3)$$

The first part models the single-phase convective heat transfer during the boiling. The quenching heat flux is generated by the liquid flow that fills the space after the bubble departure. The evaporative heat flux is the energy for generating the vapour bubble next to the wall [30].

$$q''_{CB} = h_{lw}(T_w - T_l^{cell})(1 - A_b) \quad (4)$$

$$q''_{QB} = 2\pi^{-0.5} A_b (f \lambda_l \rho_l C_{p,l})(T_w - T_l^{cell}) \quad (5)$$

$$q''_{EB} = \frac{\pi}{6} D_w^3 f N_w \rho_v L \quad (6)$$

where, h_{lw} is the convective heat transfer coefficient derived from the log law of the temperature wall function of turbulence flow. The nucleate bubble cover area is calculated from $A_b = \min(1, 1.2\pi \times e^{(-Ja_{sub}/80)} N_w D_w^2)$, where $Ja_{sub} = \rho_l C_{p,l} \Delta T_{sub} / \rho_v L$ [31]. The bubble departure diameter, D_w , is determined with system pressure on the basis of Unal's correlation [32]; the bubble departure frequency is modelled with $f = \sqrt{4g(\rho_l - \rho_v) / 3\rho_l D_w}$ [33]; and the nucleate site density is calculated from $N_w = 210^{1.805} (T_w - T_{sat})^{1.805}$ [34]. Comprehensive mathematical descriptions could be found in Ref. [3], in which it was tested that the model was able to predict the water boiling heat transfer in good precision for jet impingement configuration.

3.2.2. Mesh generation

In order to conduct the numerical simulation, a high quality block based hexahedral mesh is generated from ANSYS-Icem V16.1, as shown in Fig.3. Given that the geometry is relatively complex, the structured mesh is difficult to create. A hexagon-shaped topology is used in the zone of stagger arrayed tubes, whereas the O-grids are made around the small tubes, the inlet of water, and the edge of the cylindrical heat exchanger. The thickness of the first cell next to the wall, on the water side, is relatively large due to the requirements of RPI boiling model. A desired thickness should keep the y^+ in the range between 30 and

300. However, the y^+ on the air side must be less than 1 so that the wall function shall not be activated. After travelling through the small tubes, higher velocity air jets and larger vortexes are generated. Thus, the lateral straight air channel is prolonged to be 130 mm to prevent the reverse flow. Only half of the domain is simulated due to the symmetrical geometry. A mesh-independent result could be reached if the meshes of boiling water domain, solid domain and hot gas domain have 747,520 cells, 2,071,760 cells and 5,523,700 cells, respectively. It has been tested that doubling the mesh quantities induces a change of averaged wall heat flux within 1.5%.

3.2.3. Heat transfer coupling method

Once the multi-phase solver of ANSYS-Fluent is activated, it must apply to both fluid domains. Therefore, three different inter-phase interactions need to be defined, in which the air-water, air-steam phase interactions should be closed. Solving the three-domain conjugation case is not only resource-consuming, but also difficult to converge. An alternative solution is separating the problem into two case files and solving them separately. As shown in Fig.4, Case 1 includes the water-steam boiling flow domain and the solid conduction domain, whereas Case 2 consists of the solid conduction domain and the single-phase air convection. In order to exchange the information between two cases efficiently, the solid domain is overlapped. In other words, the water-solid two-domain conjugation problem is solved in one case, whereas the air-solid two-domain conjugation is solved in another. Given that the boiling heat transfer is sensitive to wall temperature but not wall heat flux, the wall heat flux profile of the solid-air interface in Case 2 is exported and set as the boundary condition in Case 1. In the same way, the wall temperature profile of the water-solid interface in Case 1 is exported from the results file and then imposed on Case 2 as the thermal boundary condition. An ANSYS-Fluent GUI code is developed to link the two cases together and exchange the information in between. It is able to load the

case and data files, set the imported profile as the boundary conditions, solve the case and export the profile for the other case. The two cases are exchanging information after every 100 iterations. Note that the heat flux profile on the solid-air interface is exchanged during the iterations, the predicted wall temperature profiles in the two cases are compared after convergence. It is found that the averaged wall temperature difference is less than 0.1% in the two cases, suggesting that the method introduced above works well.

4. Results and discussion

4.1. Experimental results

The experiments were conducted at three different water inlet velocities and three air inlet mass flow rates. Fig.5 demonstrates the variations of the transferred net heat energy through the heat exchanger. Increasing the air inlet temperature results in a linear increase in the net heating power, Q_w . Besides, the net heating power increases with an increase in air inlet velocity, but is independent of water inlet velocity. It suggests that the thermal resistance of the three-domain conjugation heat transfer is almost kept constant. The total thermal resistant is mainly composed of the boiling flow thermal resistance, conductive thermal resistance, and single-phase convective thermal resistance. Increasing the velocity of water does not change the boiling heat transfer coefficient, and therefore have limited influence on total resistance. However, varying the air inlet mass flow rate significantly impacts the single-phase convective thermal resistance. Increasing the air velocity results in a higher convective heat transfer rate and in turn reduces the thermal resistance and increases the heat transfer.

Fig.6 presents the variation of measured wall temperatures as a function of water and air inlet velocities. Again, it shows that the measurements are independent of water velocity in all the cases. The water inlet velocity slightly influences the wall temperatures in the low

air velocity cases ($m_{air}=2$ g/s). The measured wall temperature around the tubes, $T_{w,1-3}$, are close especially in the high air velocity cases. The measured $T_{w,1}$ is slightly lower than $T_{w,2}$ and $T_{w,3}$, because the flow temperature in the inlet zone has a 2.5 °C subcooling, but it would become saturation when it travels to the downstream. Therefore, $T_{w,2}$ and $T_{w,3}$ are almost same. $T_{w,4}$ is always significantly lower than the other three temperatures. The maximum temperature difference in the core region ($T_{w,3}-T_{w,1}$) is only around 17.7% or 24.6% of the core-to-peripheral temperature difference ($T_{w,3}-T_{w,4}$) in the high or low air inlet velocity cases. As demonstrated in Fig.2, the first three thermocouples are placed in the vicinity of the tube array whereas the last one is close to the edge of outlet. It is noteworthy that the measured wall temperature is the external wall (the air-solid interface) but not the water-solid interface. Even though the wall temperature on the water boiling side is quite uniform, the heat flux variation across the surface may induce significant external surface temperature gradient. On the air convection side, it is understandable that heat transfer in the vicinity of the tubes should be higher than the edge due to a strong turbulence induced by the sudden convergence and expansion. The heat transfer on the edge of the main channel is low due to the stagnation vortexes. Therefore, the surface could be separated into two zones due to the different heat transfer characteristics, the high heat flux core and the low heat flux peripheral region.

The averaged wall temperature variation against the net wall heat power is shown in Fig.7. Again, the water velocity has negligible effect. With a given heating power, decreasing the air inlet mass flow rate increases the wall temperature. However, the two lower m_{air} experiments present a limited difference. It suggests that flow inside the heat exchanger steps into boiling in the low m_{air} experiments while it is mainly in single-phase convection regime at high m_{air} experiments. To reach the same Q_w , the air inlet temperature must be higher in the experiment of low m_{air} . For example, when the net

wall heating power is around 70 w, the inlet air temperature is 165 °C or 200 °C in the flow rate at 4 g/s and 2 g/s, respectively. Once the boiling is established, the boiling thermal resistance does no longer change, therefore, the wall temperatures under a given heat flux should be same. It indicates that a same heat energy could be transferred with a combination of low temperature and high velocity or vice versa.

The distributions of heat transfer coefficient, h , are shown in Fig.8. The heat transfer coefficients on the air side and water side surface are defined as,

$$h_{air} = \frac{Q_w}{A_{air}(\bar{T}_{w,air} - T_{f,air})} \quad (7)$$

$$h_{wat} = \frac{Q_w}{A_{wat}(\bar{T}_{w,wat} - T_{in,wat})} \quad (8)$$

where $\bar{T}_{w,air}$ is the average of measured four surface temperatures on the air-solid interface, and $\bar{T}_{w,wat}$ is the estimated mean surface temperature on the water-solid interface. The estimation is made by applying the one-dimensional conduction equation in the solid domain as $\bar{T}_{w,wat} = \bar{T}_{w,air} - \bar{\delta}Q_w/\lambda_{sol}$, in which $\bar{\delta}$ is the mean thickness and λ_{sol} is the thermal conductivity of the solid domain. Besides, the air flow temperature, $T_{f,air}$, is evaluated with $T_{f,air} = (T_{in,air} + T_{out,air})/2$, and the heat transfer areas, A_{air} and A_{wat} , are evaluated with the exposed surfaces on both side of the heat exchanger. Given that the geometry is complex and the conduction is three-dimensional, the mean wall temperature inside the chamber reported in Fig.6 is just an estimation. It shows that the boiling heat transfer is a function of wall temperature (or wall heat flux) whereas the single-phase convective heat transfer coefficient is almost independent of wall temperature. It shows that the water side heat transfer coefficient is around three times higher than that of the air side. Varying the water velocity does not change the heat transfer coefficient on both sides. Although the maximum air temperature and the measured wall temperature are higher than the

saturated temperature of water at around 130 °C and 55 °C respectively, it seems that the wall superheat inside the chamber is not high. The boiling inside the chamber should be still in the incipient boiling regime. The fully developed nucleate boiling regime has not been reached yet. What is more, it is possible that the boiling is only initiated on a part of the inner surface, which means a co-existence of single-phase convection and boiling inside the water flow domain.

4.2. Model validation

Consequently, the numerical simulations are performed in order to explain the experimental observations. However, the validation of the simulation is conducted before the analysis of the predicted flow field and heat transfer. In the simulations, the fluid inlet and outlet boundary conditions are kept exactly same with the experiments. The water inlet boundary conditions are set to be 0.4 m/s in velocity, 97 °C in temperature where the air inlet is set to be 0.004 kg/s in mass flow rate, 116 °C to 209 °C in temperature. The outlet pressures of both cases are set to be 1 atm. The mean wall heat flux and temperature on the solid-air interface are compared in Figs.9(a) and (b). It indicates that the predicted wall heat flux is around 10% higher than the measured ones, however, they are generally agreeable. The simulated wall temperatures are lower than the measured ones. The presented wall temperatures are the average wall temperature of the core area (the average temperature of $T_{w,1}$, $T_{w,2}$ and $T_{w,3}$) both in the simulations and in the experiments. Since the compared wall temperature is that of air-solid interface, a good agreement of the wall heat flux data suggests that the air convective heat transfer is well predicted. It is inferred that the discrepancy may come from the thermal resistance in the experimental test section. The small tubes are welded on two flat plates, and then the plates are welded on the frame. There are contact thermal resistances existing between the contacted surfaces. However, all the welded parts are considered as one piece in the simulation. Therefore,

the thermal resistance would result in the discrepancy of the external wall temperatures especially in the high wall heat flux scenarios.

In the boiling flow, the void fraction of each phase is the important parameter. The RPI boiling model is able to predict the distribution of the volume fraction, however the experimental measurements is challenging. Even though the wall heat flux and wall temperature could be precisely predicted, there might be a significant discrepancy between the simulated volume fraction and the real one. As the stainless-steel materials for heat exchanger, it is difficult to measure the distribution of the volume fraction inside the chamber. Therefore, attempts have been made by capturing the bubbly flow pattern at the outlet of the test section with a high-speed camera (see Fig.9(c)). The predicted mean outlet vapour volume fraction, Φ_{sim} , is marked below each image. A transparent glass tube, with an inner diameter of 6.5 mm, is connected to the outlet of the test section, before which the chamber is insulated to avoid the heat loss induced condensation. A Phantom V711 high-speed camera (equipped with a 65 mm f2.8-16 Canon lens) and a LEICA CLS150 LED lamp are placed in front and behind of the glass tube to record the shadow image of the bubbles. The frame rate, resolution and the exposure time are of 1000 fps, 800×600 , and 5 microseconds, respectively. From the captured images, it can be seen that a small amount of bubbles are flowing through the tube when the air inlet temperature is 166 °C, where the predicted Φ_{sim} is 0.12. Increasing the air inlet temperature promotes the bubble generation significantly. The large bubbles are observed when the air inlet temperature is 210 °C, where a Φ_{sim} of 0.51 is predicted. Once the inlet temperature increases to 229 °C, a continuous vapour cavity is captured in the experiments. The high volume fraction of the secondary phase (vapour) renders the divergence of the simulation, therefore the predicted Φ_{sim} is not available for the last case. The circular glass tube causes a refraction of the bubble images, and some of the bubbles are out of

focus, therefore, a convincing quantitative measurements could not be provided at this stage. Nevertheless, the captured bubbly flow matches well with the predicted mean outlet vapour volume fraction qualitatively.

4.3. Simulation results

Fig.10 compares the wall temperature and heat flux variations on the centreline of air-solid interface in two typical cases ($T_{in,air}$ at 209 °C or 193 °C). Increasing the air inlet temperature results in the higher wall temperature and heat flux. The front surface has a higher wall temperature and heat flux compared to the back one. The distributions of those two parameters are relatively symmetrical in X-direction. However, on the back surface, the symmetrical pattern does no longer exist, since the water boiling inside the chamber is not symmetrical. The high heat transfer on the front surface attributes to the significant impingement effects and the higher air temperature. Besides, the front surface presents more significant variations along the centerline. The highest wall temperature and heat flux are observed on the edge of the tubes. On the contrary, the variations of both parameters are relatively moderate on the back surface. The wall temperature in the core region is relatively high, but it reduces sharply at the peripheral area. These results are in accordance with the experimental data that presented in Fig.4. In the experiments, the first three wall temperatures ($T_{w,1-3}$) are almost same, whereas the last one ($T_{w,4}$) is significantly lower. Although the distance between $T_{w,3}$ and $T_{w,4}$ is only 3 mm, a wall temperature difference of around 20 °C is measured. The same magnitude of temperature gradient is predicted in the simulation.

The distributions of the vapour volume fraction on different X-Y planes are illustrated in Fig.11. The liquid flow distributions on the two typical planes are also presented. Although the geometry of the water domain is symmetrical in Z-direction, the flow and heat transfer pattern is not due to the hot air flow. The vapour generation is more apparent in the

region close to the hot air inlet ($Z > 0$). The inlet water has a subcooling of around 3 °C, so the co-existence of single-phase convection and boiling is observed across the channel. On the front surface ($Z = +4.5$ mm), the vapour is generated across the entire surface. It is only generated after the third row of the tubes on the back surface ($Z = -4.5$ mm). In the middle of the channel ($Z = \pm 1.5$ mm), boiling is initiated only at the last one or two rows of tubes. The vapour bubbles are transported to the downstream and then accumulated before the outlet. A lower volume fraction of liquid results in a higher liquid speed, because the space is occupied by the vapour but the water mass fraction changes little. Therefore, it is suggested that the liquid flows faster in the vicinity of outlet.

A detailed comparison is made in Fig.12. Four different lines are drawn along the X-direction with different Z between two centre rows of tubes ($Y = 1.82$ mm). The front surface presents a higher vapour volume fraction than the back surface. The vapour volume fraction is zigzagging along X-direction due to the effects of staggered tubes. In the middle of the channel, the vapour volume fraction is notable only at the downstream half of the channel. However, it increases significantly in the outlet tube because the generated vapour get together in that region.

The air side velocity, temperature and pressure fields are plotted in Fig.13. The incoming high temperature air has a uniform temperature and impinges onto the heat exchanger, then it travels through the small tubes and expands into a large channel again. If the area of cross section reduces significantly in the small tube section, the air velocity in the tube is high. The air jets are formed after the small tubes, and the larger vortexes are generated in the lateral channel accompanied with strong flow mixing. The temperature of the downstream air flow is relatively even after a length as same as the channel diameter. The development of the thermal boundary layer inside the tube could be distinguished. Once the tube is longer, it could be anticipated that a thicker thermal boundary layer would

be formed. The averaged air temperature decreases significantly after passing through the tubes. After the small tubes, the vortexes could be generated and detached in the area between the tubes. Compared to the front air-solid interface, the rear surface has a thicker thermal boundary layer due to the low local velocity there. The convection on the rear surface is mainly caused by the jet induced reverse flow. In the area between the tubes, the vortex makes the major contribution to the thermal transportation. The temperature distribution inside the solid clearly demonstrates the aforementioned temperature gradient in the peripheral region. The impinging water jet influences the temperature distribution of the solid domain notably. The pressure field of the air flow shows the interesting results. The pressure reduces sharply at the entrance of the small tube, increases again in the middle of the tube, and then gradually decreases. It is inferred that the pressure increase inside the tube is induced by the cooling effect made by the water on the other side of the solid domain. The pressure patterns in different tubes vary significantly, indicating that the boiled flow and the hot air flow are influencing each other. Therefore, solving the conjugation heat transfer problem is reasonable and necessary in this case.

5. Conclusions

A three-domain conjugation heat transfer problem involving a two-phase water boiling heat transfer, a solid conductive heat transfer and single-phase air convective heat transfer, was experimentally investigated and then numerically simulated. The simulation was performed based on the commercial simulation software ANSYS-Fluent and GUI code, which was able to predict the boiling involved three-domain conjugation heat transfer problem. The experimental results showed that the wall temperatures on the solid-air interface and the transferred energy were independent of water inlet velocity but significantly depended on the air inlet mass flow rate. The wall temperature in the centre core area (tube array

region) was relatively uniform, whereas a huge temperature gradient was measured at the peripheral area. The maximum temperature difference in core region was around 17.7% or 24.6% of the core-to-peripheral temperature difference in the high or low air inlet velocities cases. The thermal resistance of the air side convection made the major contribution to the heat transfer. The predicted wall temperature and heat flux agreed well with the measurements. In the water flow domain, the co-existence of single-phase water convection and the boiling was observed. The wall temperature and heat flux on the front surface were higher than the rear counterpart. Thermal boundary layers were formed inside the tubes, and a significant cross-section air temperature gradient was formed. A strong jet induced flow mixing was observed in the lateral channel. The distribution of the hot air heat transfer and the water boiling heat transfer influenced each other. It is indicated that solving the conjugation heat transfer problem was reasonable and necessary.

6. Acknowledgements

The authors would like to thank National Research Foundation, Energy Innovation Programme Office, and Energy Market Authority (EMA) of Singapore for their full support to work carried out in this paper under a research grant no. NRF2013EWT-EIRP001-017.

- [1] D.H. Wolf, F.P. Incropera, R. Viskanta, Jet Impingement Boiling, in: P.H. James, F.I. Thomas, (Eds.), *Advances in Heat Transfer*, Elsevier, 1993, pp. 1-132.
- [2] L. Qiu, S. Dubey, F.H. Choo, F. Duan, Recent developments of jet impingement nucleate boiling, *International Journal of Heat and Mass Transfer*, 89 (2015) 42-58.
- [3] L. Qiu, S. Dubey, F. H. Choo, F. Duan, Effect of conjugation on jet impingement boiling heat transfer. *International Journal of Heat and Mass Transfer*, 91 (2015) 584-593.

- [4] [M.J. Rau, S.V. Garimella, Local two-phase heat transfer from arrays of confined and submerged impinging jets. International Journal of Heat and Mass Transfer 67 \(2013\) 487-498.](#)
- [5] [N.M. Dukle, D.K. Hollingsworth, Liquid crystal images of the transition from jet impingement convection to nucleate boiling part II: Nonmonotonic distribution of the convection coefficient. Experimental Thermal and Fluid Science 12 \(1996\) 288-297.](#)
- [6] [K. Wang, S. Sanders, S. Dubey, F.H. Choo, F. Duan, Stirling cycle engines for recovering low and moderate temperature heat: A review, Renewable and Sustainable Energy Reviews, 62 \(2016\) 89-108.](#)
- [7] [D. Sun, K. Wang, Y. Guo, J. Zhang, Y. Xu, J. Zou, X. Zhang, CFD study on Taconis thermoacoustic oscillation with cryogenic hydrogen as working gas, Cryogenics, 75 \(2016\) 38-46.](#)
- [8] [Y. Li, Z. Liu, G. Wang, L. Pang, Experimental study on critical heat flux of high-velocity circular jet impingement boiling on the nano-characteristic stagnation zone. International Journal of Heat and Mass Transfer 67 \(2013\) 560-568.](#)
- [9] [Y. Li, Y. Chen, Z. Liu, Correlations for boiling heat transfer characteristics of high-velocity circular jet impingement on the nano-characteristic stagnation zone. International Journal of Heat and Mass Transfer 72 \(2014\) 177-185.](#)
- [10] [Y. Qiu, Z. Liu, Critical heat flux in saturated and subcooled boiling for R-113 jet impingement on the stagnation zone. Applied Thermal Engineering 25 \(2005\) 2367-2378.](#)
- [11] T. Muszynski, D. Mikielewicz, Comparison of heat transfer characteristics in surface

cooling with boiling microjets of water, ethanol and HFE7100, *Applied Thermal Engineering*, 93 (2016) 1403-1409.

- [12] [D. Guo, J.J. Wei, Y.H. Zhang, Enhanced flow boiling heat transfer with jet impingement on micro-pin-finned surfaces, *Applied Thermal Engineering*, 31 \(2011\) 2042-2051.](#)
- [13] [S. Mitra, S.K. Saha, S. Chakraborty, S. Das, Study on boiling heat transfer of water/CTiO₂ and water/CMWCNT nanofluids based laminar jet impingement on heated steel surface, *Applied Thermal Engineering*, 37 \(2012\) 353-359.](#)
- [14] [S. Das, D.S. Kumar, S. Bhaumik, Experimental study of nucleate pool boiling heat transfer of water on silicon oxide nanoparticle coated copper heating surface, *Applied Thermal Engineering*, 96 \(2016\) 555-567.](#)
- [15] [M. Dharmendra, S. Suresh, C.S. Sujith Kumar, Q. Yang, Pool boiling heat transfer enhancement using vertically aligned carbon nanotube coatings on a copper substrate, *Applied Thermal Engineering*, 99 \(2016\) 61-71.](#)
- [16] [X. Fang, R. Wang, W. Chen, H. Zhang, C. Ma, A review of flow boiling heat transfer of nanofluids, *Applied Thermal Engineering*, 91 \(2015\) 1003-1017.](#)
- [17] [R. Cardenas, V. Narayanan, Heat transfer characteristics of submerged jet impingement boiling of saturated FC-72. *International Journal of Heat and Mass Transfer* 55 \(2012\) 4217-4231.](#)
- [18] [Z. Zhao, Y. Peles, M.K. Jensen, Water jet impingement boiling from structured-porous surfaces. *International Journal of Heat and Mass Transfer* 63 \(2013\) 445-453.](#)
- [19] [F.J. Hong, C.Y. Zhang, W. He, P. Cheng, G. Chen, Confined jet array impingement boiling of subcooled aqueous ethylene glycol solution. *International Communications in Heat and Mass Transfer* 56 \(2014\) 165-173.](#)

- [20] [D.T. Vader, G.M. Chrysler, R.C. Chu, R.E. Simons, Experimental investigation of subcooled liquid nitrogen impingement cooling of a silicon chip. Components, Packaging, and Manufacturing Technology, Part A, IEEE Transactions on 18 \(1995\) 788-794.](#)
- [21] [S. Ndao, Y. Peles, M.K. Jensen, Experimental investigation of flow boiling heat transfer of jet impingement on smooth and micro structured surfaces. International Journal of Heat and Mass Transfer 55 \(2012\) 5093-5101.](#)
- [22] [P. Zhang, G.H. Xu, X. Fu, C.R. Li, Confined jet impingement of liquid nitrogen onto different heat transfer surfaces. Cryogenics 51 \(2011\) 300-308.](#)
- [23] [E.A. Browne, G.J. Michna, M.K. Jensen, Y. Peles, Microjet array single-phase and flow boiling heat transfer with R134a. International Journal of Heat and Mass Transfer 53 \(2010\) 5027-5034.](#)
- [24] [W. Timm, K. Weinzierl, A. Leipertz, Heat transfer in subcooled jet impingement boiling at high wall temperatures. International Journal of Heat and Mass Transfer 46 \(2003\) 1385-1393.](#)
- [25] [A.M.T. Omar, M.S. Hamed, M. Shoukri, Modeling of nucleate boiling heat transfer under an impinging free jet. International Journal of Heat and Mass Transfer 52 \(2009\) 5557-5566.](#)
- [26] D. Wang, E. Yu, A. Przekwas, A computational study of two phase jet impingement cooling of an electronic chip, Fifteenth Annual IEEE Semiconductor Thermal Measurement and Management Symposium, (1999) 10-15.
- [27] [S. Narumanchi, A. Troshko, D. Bharathan, V. Hassani, Numerical simulations of nucleate boiling in impinging jets: Applications in power electronics cooling. International Journal of Heat and Mass Transfer 51 \(2008\) 1-12.](#)

- [28] [S. Abishek, R. Narayanaswamy, V. Narayanan, Effect of heater size and Reynolds number on the partitioning of surface heat flux in subcooled jet impingement boiling. International Journal of Heat and Mass Transfer 59 \(2013\) 247-261.](#)
- [29] ANSYS FLUENT Theory Guide, Release 16.1, August 2015, ANSYS Inc.
- [30] [R.M. Podowski, D.A. Drew, R.T.J. Lahey, M.Z. Podowski, A mechanistic model of the ebullition cycle in forced convection subcooled boiling, in: Proceedings of 8th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Kyoto, Japan, 1997, 1535–1542](#)
- [31] [V.H. Del Valle, D.B.R. Kenning, Subcooled flow boiling at high heat flux. International Journal of Heat and Mass Transfer 28 \(1985\) 1907-1920.](#)
- [32] H.C. Unal, Bubble diameter, maximum bubble growth time and bubble growth rate during subcooled nucleate boiling of water upto 17.7 MW/m², International Journal of Heat and Mass Transfer 19 (1976) 643–649.
- [33] [R. Cole, A photographic study of pool boiling in the region of the critical heat flux. AIChE journal 6 \(1960\) 533-538.](#)
- [34] [M. Lemmert and L. M. Chawla. Influence of flow velocity on surface boiling heat transfer coefficient in Heat Transfer in Boiling. E. Hahne and U. Grigull, Eds., Academic Press and Hemisphere, 1977, pp. 237-247.](#)

Table captions

Table 1: Physical properties of working fluids and the flow parameters.

Figure captions

Figure 1: (a) The description of the three-domain conjugation heat transfer problem which includes a two-phase boiling flow zone, a solid conduction zone and a single-phase convection zone. The hot air travels through the small tubes and boils the water on the other side of the solid cavity. (b) The configuration of the tube array. (c, d) The dimensions of the tubes.

Figure 2: The schematic of the experimental setup.

Figure 3: The 3D views (a, b) and the 2D views (c, d) of the generated hexahedral structured mesh.

Figure 4: The realization of the three-domain conjugation simulation in the Fluent V16.1 solver. Two cases are solved separately whereas the information on the solid-fluid interface is exchanged.

Figure 5: The variations of net heating power at different water inlet velocities and hot air inlet mass flow rates.

Figure 6: The surface temperature variations. The surface temperature is relatively even in the core region ($T_{w,1-3}$) whereas a high gradient is observed at the peripheral region (between $T_{w,3}$ and $T_{w,4}$).

Figure 7: The comparison of averaged wall temperature variations at different flow inlet velocities.

Figure 8: The variations of heat transfer coefficients on the air side and the water side walls ($\dot{m}_{air}=6$ g/s).

Figure 9: The comparison of the measured (a) heating power and (b) wall temperature against the numerical predictions. (c) The captured bubbly flow and the predicted mean vapour volume fraction (Φ_{sim}) at the outlet of the test section.

Figure 10: The distributions of wall temperature and heat flux on the centreline of air-solid interface ($Z = \pm 7.5$ mm).

Figure 11: The vapour volume fraction distributions in different X-Y planes (a) $Z=\pm 4.5$ mm, (b) $Z=\pm 3.5$ mm, (c) $Z=\pm 2.5$ mm, (d) $Z=\pm 1.5$ mm, and (e) the associated liquid flow velocity field ($T_{in,air}=193^\circ\text{C}$). (f) The coordination system.

Figure 12: The vapour volume fraction variations on the selected lines ($T_{in,air}=193^\circ\text{C}$).

Figure 13: The simulation results of air-solid case, which shows (a) the air temperature distribution and the associated liquid flow velocity field, (b) the temperature distribution in the solid domain, and (c) the pressure distribution of the air flow ($T_{in,air}=193^\circ\text{C}$).

Table 1: Physical properties of working fluids and the flow parameters.

	Water		Air
	Liquid	Vapour	
Reference Pressure (kPa)	101		101
Saturation temperature ($^{\circ}\text{C}$)	100		-
Surface tension (N/m)	0.059		-
Latent heat (kJ/kg)	2256		-
Density (kg/m^3)	958.4	0.598	ideal gas
Thermal conductivity ($\text{W}/\text{m}\cdot^{\circ}\text{C}$)	0.679	0.0251	0.0314
Viscosity ($\text{uPa}\cdot\text{s}$)	281.8	12.27	23.06
Specific heat ($\text{J}/\text{kg}\cdot^{\circ}\text{C}$)	4216	2080	1009
Inlet diameter (mm)	6		38
Inlet temperature ($^{\circ}\text{C}$)	97.5		100 to 230
Inlet velocity (m/s)	0.1 to 0.4		1.86 to 7.52
Inlet MFR (g/s)	2.7 to 10.8		2.0 to 6.0
Inlet Reynolds number	2040 to 8161		5214 to 2906
In-tube air velocity (m/s)	-		25.0 to 100.7
In-tube air Reynolds number	-		1434 to 2573

Figure 1: (a) The description of the three-domain conjugation heat transfer problem which includes a two-phase boiling flow zone, a solid conduction zone and a single-phase convection zone. The hot air travels through the small tubes and boils the water on the other side of the solid cavity. (b) The configuration of the tube array. (c, d) The dimensions of the tubes.

Figure 2: The schematic of the experimental setup.

Figure 3: The 3D views (a, b) and the 2D views (c, d) of the generated hexahedral structured mesh.

Figure 4: The realization of the three-domain conjugation simulation in the Fluent V16.1 solver. Two cases are solved separately whereas the information on the solid-fluid interface is exchanged.

Figure 5: The variations of net heating power at different water inlet velocities and hot air inlet mass flow rates.

Figure 6: The surface temperature variations. The surface temperature is relatively even in the core region ($T_{w,1-3}$) whereas a high gradient is observed at the peripheral region (between $T_{w,3}$ and $T_{w,4}$).

Figure 7: The comparison of averaged wall temperature variations at different flow inlet velocities.

Figure 8: The variations of heat transfer coefficients on the air side and the water side walls ($\dot{m}_{air}=6$ g/s).

Figure 9: The comparison of the measured (a) heating power and (b) wall temperature against the numerical predictions. (c) The captured bubbly flow and the predicted mean vapour volume fraction (Φ_{sim}) at the outlet of the test section.

Figure 10: The distributions of wall temperature and heat flux on the centreline of air-solid interface ($Z = \pm 7.5$ mm).

Figure 11: The vapour volume fraction distributions in different X-Y planes (a) $Z=\pm 4.5$ mm, (b) $Z=\pm 3.5$ mm, (c) $Z=\pm 2.5$ mm, (d) $Z=\pm 1.5$ mm, and (e) the associated liquid flow velocity field ($T_{in,air}=193^{\circ}\text{C}$). (f) The coordination system.

Figure 12: The vapour volume fraction variations on the selected lines ($T_{in,air}=193^{\circ}\text{C}$).

Figure 13: The simulation results of air-solid case, which shows (a) the air temperature distribution and the associated liquid flow velocity field, (b) the temperature distribution in the solid domain, and (c) the pressure distribution of the air flow ($T_{in,air}=193^{\circ}\text{C}$).