Comparison between modelled and measured heat transfer rates during the departure of a steam bubble from a solid surface

G. Giustini¹, Iljin Kim², Hyungdae Kim²

¹ Mechanical Engineering Department, Imperial College London, SW7 2AZ, United Kingdom
² Department of Nuclear Engineering, Kyung Hee University, Youngin, Republic of Korea

* Corresponding author, g.giustini12@imperial.ac.uk
zhzhboy@khu.ac.kr; hdkims@khu.ac.kr;

Abstract
This paper presents analysis of the heat transfer attendant upon the departure of a single steam bubble during pool boiling of water at atmospheric pressure. The flow of heat from a solid surface to liquid water during and immediately after bubble lift-off has been extracted from micro-scale measurements of the spatial and temporal variation of the temperature at the solid surface beneath the bubble. The numerical procedure used to extract the heat flux from the temperature variations at the solid surface has been assessed and verified, and applied to investigate the heat transfer during the bubble departure phase, and after the eventual bubble lift-off. Results confirm that fluid motion activated by a departing bubble is the cause of heat transfer enhancement. The phenomenon can be characterised as the process of rewetting, by an advancing liquid front, of a dry portion of wall area at the base of the bubble. The portion of wall area that is affected by the observed heat transfer augmentation mechanism has been found to be that of a circle of diameter roughly equal to half the bubble departure diameter.

The current measurements enabled validation of interface-capturing numerical simulation of the hydrodynamics and heat transfer of single bubble formation and departure from a surface, including conjugate heat transfer in the solid substrate. From simulation results, the spatial and temporal variation of the heat flux at the solid surface beneath the bubble has been computed and monitored during bubble departure and after the eventual bubble lift-off. Heat transfer rates at bubble departure extracted from simulation have been found in good agreement with measurements. The simulation correctly captured experimental trends and was found to give an accurate estimate of the magnitude of the flows of heat to the liquid due to the bringing of cold fluid in the vicinity of the wall caused by the bubble departure process.
1 INTRODUCTION

The boiling process is an integral part of many industrial applications such as power generation, refrigeration and cooling of processor chips, among many others [1] [2]. Nucleate boiling, i.e. the formation of steam bubbles from superheated liquid at a solid surface, provides excellent cooling, which is why these applications can achieve high power densities, yet also have relatively low component temperatures. A good understanding of nucleate boiling heat transfer is therefore vital for the development of the above applications.

A general, first-principles based nucleate boiling model does not exist yet, due to the multi-physics and multi-scale nature of the process [3]. However, practical applications require knowledge of heat transfer rates due to two-phase cooling by a boiling fluid, and therefore some physical model of the boiling process needs to be incorporated into engineering modelling of boiling flows, typically involving length scales much larger than the characteristic dimensions of the single bubbles [2]. A possible approach, widely used in Computational Fluid Dynamics (CFD) modelling of boiling flows, is to use sub-grid scale models of the micro-scale heat transfer mechanisms arising from the behaviour of a population of bubbles forming at and released from a distribution of nucleation sites at a solid surface. With this approach [4], the heat transfer associated to the boiling process is evaluated via consideration of the micro-scale heat transfer modes of a single steam bubble. Figure 1 shows a schematic of the common understanding of these single-bubble heat transfer mechanisms, adapted from Refs. [5] [6].

According to this description, heat transfer modes associated to nucleate boiling are treated separately and idealised as an ‘evaporative’ heat transfer component, representing the latent heat transfer associated to bubble formation, and a ‘single-phase’ component, representing sensible heat transfer to the liquid due to fluid motion caused by a departing bubble.

![Figure 1](image)

Our current understanding of single-bubble heat transfer modes.

Whereas it is not unnatural that the latent heat contribution due to bubble formation could be expected to be a significant fraction of the overall transport of heat during boiling, as indeed some works suggest [7], estimates of the heat flow from a solid surface to the single-phase liquid attendant upon bubble departure are somehow still uncertain. Our current understanding of the process is based on the conjecture, due to Han [5], that a departing bubble...
causes anomalously 'cold' fluid to make sudden contact with the heat transfer surface, from which heat is conducted through the liquid. A similar mechanism was proposed in Refs. [8, 9], where it was suggested that significant flow of heat is caused by the gradual, albeit quick, rewetting of the 'dry' portion of solid surface beneath a departing bubble. A better understanding of these processes is crucial for the development of computational models of boiling flows, and in particular of macroscopic two-fluid CFD techniques, which require closure relations for sub-grid scale models of wall-boiling heat transfer, and are used in a broad range of pool [10] and forced flow [11] [12] boiling calculations.

Recently, the development of micro-scale experimental [13] [14] and computational techniques [15-17] has enabled us to observe directly the process of bubble formation, and to model from first principles its hydrodynamics and heat transfer. These techniques therefore provide a well characterised opportunity to evaluate directly the flow of heat to the fluid from a solid surface near a departing bubble.

This paper is exactly in this spirit. In the first part, the growth and departure of a single steam bubble has been observed during pool boiling of water at atmospheric pressure in laboratory conditions. Post-processing of sequential measurements of the temperature distribution at the solid surface beneath the bubble enabled extracting the spatial and temporal variation of surface-normal heat flux during the bubble departure phase and following the eventual bubble lift-off. In the second part, the hydrodynamics and heat transfer of bubble formation and release, and the conjugate heat transfer in the solid substrate, have been modelled using interface-capturing numerical simulation. From simulation, the flows of heat during and after bubble departure have been computed and compared with measurements.

The current paper is structured as follows. A description of the experimental methodology, and of qualitative trends in observed bubble behaviour, are presented in Section 2. The procedure used to extract the temporally and spatially varying heat flux from a sequence of temperature measurements at the solid surface is assessed in Section 3. Direct visualization of the spatio-temporal variation of the surface normal heat flux at the solid surface beneath the departing bubble is discussed in Section 4. Section 5 presents an overview of the interface-capturing simulation methodology used in this paper to model bubble formation and release. Qualitative trends in modelled bubble behaviour are discussed in Section 6. A quantitative comparison between modelled and measured heat transfer rates to the liquid during bubble departure is presented in Section 7. Conclusions follow in Section 8.

2 THE EXPERIMENT

2.1 Description of experimental technique

The experimental technique of Jung and Kim [14] was devised to perform simultaneous measurements of various aspects of bubble formation and release in pool boiling of water at a horizontal surface. The surface is heated via Joule effect with a thin Indium Tin Oxide (ITO) film, deposited on a calcium fluoride (CaF2) substrate. The experimental setup, shown in Figure 2, consists of synchronized optical and infra-red detectors that are used to measure, respectively, the characteristic physical dimensions of a bubble, and the temperature
distribution at the solid surface beneath the bubble. In order to check the recorded shape of the bubble close to the surface, another synchronized interferometric measurement of the distribution of the vapour and liquid phases at the solid surface was considered. This enabled observation of the interaction between the growing bubble and the solid substrate, and tracking the possible formation, on the solid surface beneath the bubble, of a liquid ‘microlayer’, which is likely to occur during the early stages of bubble growth.

Figure 2
Diagram of the experimental setup used for measurements of bubble formation and release.

The infra-red detector was used to measure the temperature distribution at the underside of the thin (700 nanometers) infra-red opaque ITO film, deposited on the infra-red transparent CaF$_2$ substrate. The temperature drop between the upper and lower surfaces of the ITO film can be estimated, assuming steady heat conduction, as $\Delta T = \frac{q''_{\text{ITO}} h_{\text{ITO}}}{k_{\text{ITO}}}$, with $q''_{\text{ITO}}$ the heat flux, $h_{\text{ITO}}$ the layer thickness, and $k_{\text{ITO}}$ the layer thermal conductivity. Even assuming for the latter an anomalously low value of about 1 W/m/K recently reported for very thin layers [18], at a very high heat flux of 1 MW/m$^2$ one gets a small temperature drop of 0.7K, within the experimental uncertainty. Temporal accuracy of the measurement can be evaluated via consideration of the thermal penetration depth in the ITO, $\sqrt{\frac{k_{\text{ITO}}}{\rho_{\text{ITO}} c_{\text{ITO}}}}$, where $\rho_{\text{ITO}}$ is the density of the material and $c_{\text{ITO}}$ its thermal conductivity. Using typical values of 9000 kg/m$^3$ for the former and 800 J/kg/K for the latter, it is noted that after a short time $\tau$ of about 3.5 $\mu$s, the thermal penetration depth is already larger than the layer thickness, and that therefore the measurement of the temperature at the ITO lower surface is for all intents and purposes representative of the temperature at the ITO-fluid interface.

The procedure used to extract the heat flux in the direction normal to the solid surface relies on modelling the thermal response of the calcium substrate to the temporal variation of the temperature distribution at its upper surface (the ITO-CaF$_2$ interface) near the location where a bubble grows. A schematic of the procedure, originally developed for the analysis of droplet-
wall interactions [19], and subsequently applied to the modelling of single-bubble heat transfer [14], is shown in Figure 3. A sequence of two-dimensional measurements of the temperature distribution at the upper surface of the CaF$_2$ substrate was used as a time-dependent boundary condition of a finite volume model of unsteady heat diffusion in the CaF$_2$, from which the value of the heat flux at the surface under investigation was extracted. In the finite volume model of the CaF$_2$ substrate, all of its surfaces were treated as symmetry planes, except the one exposed to the fluid, where the above sequence of temperature fields was imposed, and the bottom plane, exposed to the atmosphere and approximated as an adiabatic boundary. The initial temperature of CaF$_2$ was taken as uniform, and equal to the average temperature of the CaF$_2$ surface under investigation. The physical time of the entire simulation was 1000 ms. To avoid the effect of the unrealistic uniform initial temperature on the heat flux analysis, a target bubble was chosen at 140 ms after a couple of bubbles were nucleated and departed.

Figure 3

Description of the procedure used to extract the surface normal heat flux from a known surface temperature distribution.

2.2 Experimental conditions

The formation and release of a single bubble during pool boiling of water at atmospheric pressure was observed at a horizontal upward-facing, nanoscopically smooth surface ($R_a = 2.58 \text{ nm}$). Conditions of the experiment are summarized in Table 1. The values of heat flux and bubble inception superheat are typical of atmospheric pressure laboratory experiments. Due to the nature of the heat flux measurements, it was necessary to use a CaF$_2$ block of thickness several times larger than the thermal penetration depth into the solid corresponding to the observed bubble release period. This precaution ensured that local measurements of the temperature near the nucleation site are independent of the thickness of the boiling substrate.
<table>
<thead>
<tr>
<th>Quantity</th>
<th>Experimental value</th>
<th>Value for substrate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Applied heat flux $q'''' [kw/m^2]$</td>
<td>144.0</td>
<td>-</td>
</tr>
<tr>
<td>Measured bubble nucleation temperature $T_{nuc} [^\circ C]$</td>
<td>110.7</td>
<td>-</td>
</tr>
<tr>
<td>Measured ebullition cycle period $\tau [s]$</td>
<td>0.09</td>
<td>-</td>
</tr>
<tr>
<td>Size of ITO heater $[mm^2]$</td>
<td></td>
<td>$5 \times 15$</td>
</tr>
<tr>
<td>Surface roughness of ITO heater $[nm]$</td>
<td></td>
<td>2.58</td>
</tr>
<tr>
<td>Density $\rho [kg/m^3]$</td>
<td>-</td>
<td>926.0</td>
</tr>
<tr>
<td>Specific heat capacity $c [J/kg/K]$</td>
<td>-</td>
<td>3129.0</td>
</tr>
<tr>
<td>Thermal conductivity $k [W/m/K]$</td>
<td>-</td>
<td>5.919</td>
</tr>
<tr>
<td>Substrate thickness $d [m]$</td>
<td>-</td>
<td>0.01</td>
</tr>
<tr>
<td>Thermal penetration depth $\delta = \frac{kr}{\sqrt{\rho c}} [m]$</td>
<td>$4.3 \times 10^{-4}$</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 1
Summary of experimental conditions.

The time history of the temperature, directly measured at the nucleation site under investigation, and of the local heat flux obtained from finite volume modelling, are shown in Figure 4 to be fairly typical of pool boiling of water in laboratory conditions ([20] [14]). In the figure, a red line indicates the time history of temperature at the nucleation site, a blue line indicates the corresponding heat flux. At 0 ms, the recorded temperature value of 110.7°C is taken as the incipient nucleation temperature. Following the initial peak in heat flux, due to latent heat transfer to the fast-growing bubble, another heat flux peak is observed around the time of bubble departure. In the following section, a discussion of the nature and cause of this second peak, way above the baseline heat flux value of 144 kW/m² (also indicated in the figure), is presented.
Illustration of conditions analyzed in the present work, showing time histories of the observed temperature and heat flux at the nucleation site under investigation. The corresponding ebullition cycle period is computed as the interval between nucleation of two successive bubbles (times 0 and ~90 ms in the figure).

2.3 Observed bubble behaviour – qualitative trends

Figure 5 shows the process of bubble formation and release observed with the above experimental technique. The observed ebullition cycle can be subdivided into three phases, namely a rapid initial bubble growth period, a subsequent departure period and a post-lift-off ‘quench’ period. The top sequence of images represents side views of the bubble from the optical camera, which have been used to track the temporal evolution of the bubble height, width and base diameter. The second sequence of images represent the laser-interferometry detection of phase distribution at the solid-fluid interface beneath the bubble from the second optical camera from below, which have been used to track the temporal evolution of the diameter of the dry wall area at the bubble base. The fringe patterns appear due to the phenomenon of interference of the coherent laser with the thin (about a micron) liquid layer ‘microlayer’ [14] and disappear in the area where the liquid layer is dried out, which permits to track the location of the diameter of the dry wall area at the bubble base. Detailed explanations of the measurement principles of the integrated total reflection and laser interferometry technique is described in ref. [14]. The third sequence of images represent the wall temperature distribution from the infrared camera and the last two rows show the boundary heat flux distribution at the solid-fluid interface for two different field resolutions.

Optical measurements of bubble behaviour enabled distinguishing three different physical situations during the bubble growth cycle, namely i) the growth a liquid microlayer beneath a bubble (early bubble growth stage, e.g. 1 ms after nucleation), ii) coexistence of microlayer and growing dry patch (towards bubble departure, e.g. 6 and 12 ms after nucleation), and finally iii) a completely dry bubble base, typical of the departure and lift-off stages.

For subsequent analyses, the heat flux distribution has been extracted along a line probe passing through the geometrical centre of the circular area identified by the recorded maximum...
base diameter (Figure 6a), and the variation along this line of wall-normal heat flux has been displayed at a sequence of times in Figure 6b.

Figure 5
From top to bottom: bubble side view from high-speed camera, laser-interferometry detection of phase distribution at the solid-fluid interface beneath the bubble, wall temperature distribution. The last two rows show the boundary heat flux distribution field at the solid-fluid interface for two different field resolutions.

Figure 6
(a) heat flux extraction line, passing through the approximate centre of the bubble base, indicated as the horizontal axis in panel (b).
3 DISCUSSION OF METHODOLOGY USED TO COMPUTE THE WALL HEAT FLUX

The numerical procedure used to extract the heat flux from sequential measurements of the temperature distribution on the solid surface is here discussed. To the best of our knowledge, a comprehensive analysis of the possible sources of error implicit in the methodology has not been presented yet for the case of nucleate boiling heat transfer measurements. The discussion that follows serves the purpose of providing reliable estimates of the accuracy and uncertainty of the current assessment of wall heat flux.

3.1 Mesh sensitivity analysis

Three-dimensional transient heat conduction in the CaF₂ substrate was simulated using the ANSYS Fluent CFD software [21]. Non-uniform structured meshes with graded cells along the thickness of the CaF₂ substrate, as shown in Figure 7a, were used for simulation of three-dimensional transient heat conduction in the solid substrate. While the width and depth of mesh were set to a fixed value of 72.5 µm, dictated by the spatial resolution of the temperature measurement using infrared camera, the cell height was refined near the boiling surface. A value of ~20 µm near the boiling surface, where steep temperature change is expected, was adopted, and then gradually increased moving toward the bottom of the CaF₂ substrate. To validate the heat flux calculation results obtained using the non-uniform mesh, a systematic mesh sensitivity analysis was performed by varying the height of uniform structured ‘test’ meshes from 145 µm to 4.53 µm indicated in Figure 7b. Details are summarized in Table 2.

Figure 8 shows the local heat flux histories at the bubble nucleation site obtained from the mesh sensitivity analysis. At time instants corresponding to bubble nucleation and bubble departure, when the local heat flux varies rapidly in time, some noticeable discrepancies in the local heat flux were observed according to the tested mesh height. The results obtained with large meshes relatively underestimated the peak heat flux, but when the mesh height is smaller than 36 µm, the computed peak heat fluxes converged to a unique value. The discrepancy in the local instantaneous heat flux at the nucleation site between successive test mesh refinements and values obtained with the reference mesh was found to be less than 3% when the mesh height is less than ~20 µm. In addition to the local instantaneous heat flux value, the mesh-dependent behavior of the spatial distribution of heat flux during the rewet phase of interest in the present study was investigated in Figure 9. Similarly, the distributions of heat flux are quantitatively consistent when the mesh height is less than ~20 µm. Therefore, it can be concluded that the heat flux distribution results obtained using the reference non-uniform structured mesh are mesh-independent and accurate.
Figure 7
The reference (a) and test (b) mesh types used in the transient conduction model.

<table>
<thead>
<tr>
<th>Height [μm]</th>
<th>Ref. (non-uniform)</th>
<th>Grid 1</th>
<th>Grid 2</th>
<th>Grid 3</th>
<th>Grid 4</th>
<th>Grid 5</th>
<th>Grid 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 (minimum)</td>
<td>145</td>
<td>72.5</td>
<td>36.25</td>
<td>18.13</td>
<td>9.06</td>
<td>4.53</td>
<td></td>
</tr>
</tbody>
</table>

Table 2
Mesh refinement levels used in grid convergence study.

Figure 8
Mesh-dependent time histories of the heat flux at the nucleation site. Panel (b) shows magnification about the time of nucleation, panel (c) shows the heat flux about the time of bubble departure.
Mesh-dependent behaviour of the spatial distribution of heat flux during the rewet phase, shown at a sequence of times during bubble departure. The heat flux has been extracted along the line indicated in Figure 6a.

3.2 Uncertainty analysis

Uncertainty of modelled heat flux values was evaluated via consideration of the fluctuations of the infrared temperature measurements and of the mesh characteristics and material properties used in the numerical model. Using the fluctuation $\Delta T$ of the temperature measurement, the heat flux can be calculated using Fourier’s law in one dimension

$$q'' = -k \frac{\Delta T}{\Delta h}$$

where $k$ is thermal conductivity of the CaF$_2$ substrate, $\Delta h$ is height of the first cell at the upper surface of the CaF$_2$ block. Uncertainty in calculating the wall heat flux can be estimated according to propagation of uncertainties [22] as

$$\frac{u(q'')}{q''} = \sqrt{\left(-\frac{1}{k} u(k)\right)^2 + \left(\frac{1}{\Delta h} u(\Delta h)\right)^2 + \left(-\frac{1}{q'' \Delta h} u(\Delta T)\right)^2}$$

The thermal conductivity of the CaF$_2$ substrate was measured by LFA 447 NanoFlash™ (NETZSCH) [23]. The average value of the three measurements at 100°C was 5.919 W/m/K and the associated measurement uncertainty was about 6%. The mesh size ($\Delta h$) is set equal to 20 microns for the reference mesh used throughout this paper, and its uncertainty is zero. Lastly, the temperature measurement data by the infrared camera used in the present study exhibits a characteristic random fluctuation error within ±0.1°C [24], which determines the uncertainty for the wall heat flux calculation. The uncertainties and input values used for heat flux uncertainty evaluation are summarized in Table 3.

The resulting uncertainty of the local wall heat flux calculation was found to be about 30 kW/m$^2$, which is about 20% of the average heat flux applied to the heater, 6.4% of the peak wall heat flux during the bubble formation process, and 11.5% of the peak heat flux during bubble departure process which is the main focus of this study. Figure 10 summarises the uncertainty analysis results.
Discrepancy in local instantaneous heat flux values at the nucleation site between successive test mesh refinements and values obtained with the reference mesh. The discrepancy, computed as $\varepsilon = \frac{q''_{\text{ref}} - q''_{\text{grid}#}}{q''_{\text{ref}}}$, has been evaluated at the time of nucleation (0 ms) and at bubble lift-off (20 ms). $q''_{\text{ref}}$ indicates the heat flux value computed with the reference mesh, $q''_{\text{grid}#}$ corresponds to the test mesh, and # is the test mesh number.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k$ [W/m/K]</td>
<td>5,919</td>
<td>0.355</td>
</tr>
<tr>
<td>$\Delta h$ [µm]</td>
<td>20</td>
<td>-</td>
</tr>
<tr>
<td>$\Delta T$ [K]</td>
<td>-</td>
<td>0.1</td>
</tr>
<tr>
<td>$q''_{\text{input}}$ [kW/m²]</td>
<td>144</td>
<td>30.8</td>
</tr>
<tr>
<td>$q''_{\text{formation}}$ [kW/m²]</td>
<td>~1300</td>
<td>83.4</td>
</tr>
<tr>
<td>$q''_{\text{departure}}$ [kW/m²]</td>
<td>~300</td>
<td>34.6</td>
</tr>
</tbody>
</table>

Table 3
Uncertainties and input values used for heat flux uncertainty evaluation

4 VISUALIZATION OF THE SPATIO-TEMPORAL VARIATION OF HEAT FLUX AT THE SOLID SURFACE DURING BUBBLE DEPARTURE

4.1 Phase distribution on heat transfer surface

The distribution of the liquid and vapour phases on the heat transfer surface during various stages of bubble formation is shown in Figure 11. The process of interest is the behaviour of the dry portion of wall area, characterized by an increase in dry area during bubble growth,
followed by its decrease during the departure phase. On the left extremity of panel (a), showing the phase distribution typical of the early stages of bubble growth, the fringe pattern typical of the microlayer is clearly visible, leaving only a very small central dry region. As the microlayer evaporates, the dry patch grows in size, as indicated by the image taken at 6 ms after bubble nucleation, a behaviour that continues until the microlayer has been almost completely depleted, as visible after 12 ms. Eventually, after 15 ms, the microlayer has disappeared completely, and the bubble base is left entirely dry. In panel (b), the decrease in bubble base area size occurring during the bubble departure process is illustrated until the time of bubble lift-off, taking place approximately at 20 ms after nucleation.

![Figure 11](image)

Distribution of phases in contact with the solid surface from interferometric measurements.
(a) typical phase distribution during various phases of bubble formation, (b) during bubble departure, showing decrease in size of the dry bubble base.

### 4.2 Radial heat flux profiles

Enhanced cooling due to the action of the advancing liquid ‘rewet’ front is clear from the trends shown in Figure 12. Panels (a) and (b) show magnification of the instantaneous planar heat flux distribution during and after bubble departure. Panels (c) and (d) show corresponding radial distributions of the heat flux values, from which the baseline heat flux of 144 kW/m² has been subtracted. The magnitude of the peak heat flux is seen increasing with time as the liquid front covers the wall dry area; later on, following bubble departure, the heat flux decays. This behaviour indicates that the largest flow of heat to the liquid occurs during progressive rewetting of the wall dry area, whereas the following quench of the surface due to liquid set in motion by the departing bubble results in comparatively smaller heat flow rates. This is in
qualitative agreement with earlier results presented in ref [17]. The magnitude (±30 kW/m²) of the heat flux experimental error, well below the observed heat flux peak values, is noted. (For clarity, error bars are not shown in the figure).

![Figure 12](image)

(a) planar and (c) radial heat flux distribution at the solid surface during the process of bubble detachment, or ‘rewet’ heat flux.
(b) planar and (d) radial heat flux distribution at the solid surface after bubble lift-off.

Another quantity of interest that was extracted from synchronized measurements is the extension of the portion of wall area that is affected by heat transfer augmentation during and following bubble departure. It is recalled that the extension of the dry area beneath a bubble decreases in time as the instant of bubble lift-off is approached, as indicated in Figure 11, due its rewetting by an advancing liquid front. It is therefore important to relate the extension of the heat transfer augmentation zone to the maximum extension of the bubble dry base area and to the bubble size at departure. From Figure 13, it is clear that the heat transfer augmentation area does not exceed that of a circle of radius equal to approximately half the observed bubble departure radius, which is in qualitative agreement with the analysis presented in ref. [17]. This is contrast with extant models of the bubble departure process [25] [5] [4], typically assuming larger areas of influence, for which the radial extent of the area of influence of the heat transfer mode here under investigation could vary between one and two bubble departure radii.
Direct visualization of the area of influence of heat transfer augmentation beneath a departing bubble. The instant of bubble lift-off was recorded between 19 and 20 ms after bubble nucleation.

5 OVERVIEW OF INTERFACE CAPTURING SIMULATION METHODOLOGY

The interface-capturing methodology embodied in the open-source PSI-BOIL code [26] was used to model the hydrodynamics and heat transfer of bubble formation and release from a surface, including conjugate heat transfer in the solid substrate beneath a bubble. As an initial condition of the simulation, a small ‘seed’ of vapour is initialized, and its growth on the solid surface due to evaporation is tracked. A sketch of the indicative bubble shape, temperature distribution in the solid, and heat flux distribution at the solid-fluid interface is shown in Figure 14a, illustrating typical simulated behaviour during the early stages of bubble growth. It is noted how solving for conjugate heat transfer in the solid enables tracking the variation of the temperature distribution in the solid due to heat transfer to the bubble and to the surrounding liquid.

An important aspect required to be captured in order reproduce experimental trends in bubble behaviour concerns the early stages of bubble growth. In pool boiling of water at atmospheric pressure, an evaporating thin liquid film is typically present beneath the bubble during the early stages of its growth. Microlayers were observed forming, evaporating and eventually completely disappearing (thus leaving a ‘dry’ spot at the solid surface beneath a bubble) in the current set of experiments, as discussed in the preceding text, and it was found necessary to take into account the associated evaporation rates in the computational model. This is done with an algebraic sub-grid scale model [27] embedded in the PSI-BOIL code. The model, as sketched in Figure 14b, tracks the depletion due to evaporation of the thin liquid film assuming that heat is transferred via conduction between the upper and lower surface of the film, and it
also tracks the temporal variation of the radial extension of the film, and its eventual disappearance. In this work, the temperature of the lower surface of the microlayer (the solid-liquid interface) is computed from solution of the solid conjugate heat transfer problem, and the temperature of the upper surface of the microlayer (the liquid-vapour interface) is taken as the saturation temperature at the externally imposed pressure. As shown in Figure 14b, the model allows tracking the temporal variation of the radius of the bubble base \( r_{bb} \) computed from the interface capturing solution in the fluid domain and of the dry patch created by microlayer depletion \( r_{w} \) computed by the algebraic sub-grid scale model).

Figure 14
Illustrating the simulation methodology used to predict bubble formation and departure [17]. Panel (a) shows indicative qualitative trends in bubble shape, temperature distribution in the solid substrate, and heat flux radial variation at the solid surface during the early stages of bubble growth. During this initial bubble growth phase, vapour generation from the coupled solid substrate / microlayer thermal model (b) contributes to bubble expansion.

Following an initial growth phase, the microlayer dries out completely, leaving a dry spot beneath the bubble, as confirmed by the current experiments. The ref. [17] methodology allows computing, during this bubble departure stage, the decrease in dry-spot radius, and its being rewetted by an advancing liquid front, shown in Figure 15a, eventually leaving the solid surface entirely in contact with liquid, as indicated in Figure 15b. An overview of the model
equations used to predict the ebullition cycle outlined in the preceding text is presented in the following section.

![Figure 15](image)

During the bubble departure phase (a), the entirety of the bubble base is dry (‘dry patch’), and rewetting of the dry patch by an advancing liquid front occurs. After bubble departure (b), the solid substrate is entirely in contact with liquid.

### 5.1 Model equations

The open source PSI-BOIL code [26] is used to solve the incompressible Navier-Stokes equations with a projection method [28] on staggered Cartesian grids. Indicating vector quantities in bold, the momentum equations are formulated as in Ref [16]:

\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \rho [\nabla \cdot (\mathbf{uu}) - \mathbf{u} (\nabla \cdot \mathbf{u})] = -\nabla p + \nabla \cdot (\mu [\nabla \mathbf{u} + (\nabla \mathbf{u})^T]) + \rho g + \mathbf{f} \tag{3},
\]

where \( \rho \) is the fluid density, \( \mathbf{u} \) its velocity, \( p \) the pressure, \( \mu \) the dynamic viscosity, \( g \) the gravitational acceleration vector and \( \mathbf{f} \) the surface tension force. A generic fluid property, say the density \( \rho \), varies as an arithmetic average using the liquid volume fraction \( \alpha \) as weight, \( \rho = \rho_\alpha \alpha + \rho_v (1 - \alpha) \). The buoyancy force \( \rho g \) is modelled via the Boussinesq approximation, and the surface tension force, including wall-adhesion in the near-wall cells, is treated as a volume force and computed with the Continuum Surface Force (CSF) method of Ref. [29]. This approach computes the surface tension force at a solid surface from the unit normal vector to the vapour-liquid interface (computed from the volume fraction distribution in the interface region) and contact angle value (supplied as input to the model).

The interface location is tracked with an advection equation for the liquid volume fraction \( \alpha \) [30]:

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) = -\dot{m} \frac{1}{\rho_l} \tag{4},
\]

where \( \dot{m} \) is the interphase mass transfer rate, and the location of the vapour-liquid interface is identified by the condition \( \alpha = 0.5 \).

In the PSI-BOIL, pressure-velocity coupling algorithm, the velocity divergence is linked to the interfacial mass transfer rate via a continuity equation:
\[ \nabla \cdot \mathbf{u} = \dot{m} \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \quad (5). \]

The mass transfer rate is computed, in the cells containing the vapour-liquid interface, via evaluation of the liquid- and vapour-side temperature gradients [16], under the assumption that the interface is at the saturation temperature at the externally imposed pressure (that is, in conditions of interest, 100°C at 101325 Pa):

\[ \dot{m} = \frac{k_t (\nabla T)_l - k_t (\nabla T)_v}{h_{fg}} \frac{S_{int}}{V_{cell}} \quad (6), \]

where \( n \) is the interface unit normal (pointing towards the liquid), \( k \) the thermal conductivity, \( h_{fg} \) the latent heat of evaporation and \( S_{int} \) is the portion of bubble curved surface contained in a computational cell of volume \( V_{cell} \). The temperature gradients are computed from a temperature distribution resulting from solution of an enthalpy balance equation:

\[ \rho c \left[ \frac{\partial T}{\partial t} + \nabla \cdot (T \mathbf{u}) - T (\nabla \cdot \mathbf{u}) \right] = \nabla \cdot (k \nabla T) \quad (7), \]

where \( c \) is the specific heat capacity.

The same thermal model is used in the solid, with the appropriate thermophysical properties, and flow velocities set to zero.

The mass transfer rate due to microlayer evaporation [27], added to the source term in the continuity equation, is computed from the rate of reduction of microlayer thickness \( \delta \) as

\[ \dot{m}_{micro} = -\rho_l \frac{\partial \delta}{\partial t} \frac{S_{int}}{V_{cell}} \quad (8), \]

where the initial value of the film thickness is computed as a function of the radial coordinate \( r \) from the nucleation site using the correlation by Utaka [31] (valid for water boiling at atmospheric pressure), and equal to \( \delta_0 = 4.46 \times 10^{-3} \times r \). The rate of depletion of the film is obtained assuming heat conduction between the upper and lower surface of the film, \( \frac{\partial \delta}{\partial t} = -\frac{k_t}{\rho_l h_{fg}} \frac{t_{sat} - T_{sat}}{\delta} \).

The heat sink due to latent heat transfer across the microlayer is taken into account in the layer of solid cells in contact with the fluid, and is modelled as

\[ \dot{q}'''' = h_{fg} \dot{m}_{micro} \quad (9). \]

5.2 Computational domain, boundary and initial conditions, fluid properties

The computational domain, containing both fluid and solid volumes, is a parallelepiped of suitable base side and height that have been adjusted until domain-size independent bubble behaviour was obtained.

Owing to symmetry of the problem, the three-dimensional model equations have been solved in a computational domain containing \( \frac{1}{4} \) of the bubble, with appropriate symmetry boundary conditions at the intersection of the bubble curved surface with two of the parallelepiped vertical sides. A vertical cross section of the three-dimensional domain and mesh is shown in Figure 16c, showing typical dimensions and cell sizes. The thickness of cells in the solid domain is uniform and equal to half the minimum thickness of fluid cells. For the finest mesh considered, the thickness of solid cells is equal to 15 microns.
All boundaries are adiabatic slip walls, except the solid-fluid interface, which is treated as a zero-slip surface thermally coupled with the solid, and the top boundary, through which flow of mass, momentum and energy is allowed.

As an initial condition, the vapor phase is seeded at a desired location as a small sphere of vapor, of radius equal to 1.5 times the local cell height. All flow velocities are initially set to zero. The temperature in the liquid is initialized with the correlation of Nishikawa, as in ref. [17], which imposes a drop between the incipient nucleation temperature (in this case 110.7°C) and the saturation temperature (100.0°C) over a length of approximately 1.5 mm. In the solid, a uniform temperature is initially imposed, equal to the observed nucleation temperature of 110.7°C.

Fluid properties used in the present simulations are those of water in saturation conditions at atmospheric pressure. The numerical values of the fluid properties used in the current paper are indicated in Table 4.

<table>
<thead>
<tr>
<th>Property</th>
<th>Vapour</th>
<th>Liquid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dynamic viscosity $\mu$ [Pa s]</td>
<td>$12.2 \times 10^{-6}$</td>
<td>$281.6 \times 10^{-6}$</td>
</tr>
<tr>
<td>Density $\rho$ [kg/m$^3$]</td>
<td>0.6</td>
<td>958.4</td>
</tr>
<tr>
<td>Specific heat capacity $c$ [J/ Kg/K]</td>
<td>2077.5</td>
<td>4216.6</td>
</tr>
<tr>
<td>Thermal conductivity $k$ [W/m K]</td>
<td>$24.8 \times 10^{-2}$</td>
<td>$677.8 \times 10^{-2}$</td>
</tr>
<tr>
<td>Isobaric expansion coefficient $\beta$ [K$^{-1}$]</td>
<td>$-2897.3 \times 10^{-6}$</td>
<td>$-697.0 \times 10^{-6}$</td>
</tr>
<tr>
<td>Surface tension coefficient $\sigma$ [N/m]</td>
<td>0.059</td>
<td></td>
</tr>
<tr>
<td>Latent heat of vaporization $h_{fg}$ [J/Kg]</td>
<td>$2258.0 \times 10^{3}$</td>
<td></td>
</tr>
</tbody>
</table>

**Table 4**

Fluid properties of water in saturation conditions at atmospheric pressure.

**6 MODELLING BUBBLE BEHAVIOUR**

**6.1 Optimization of numerical parameters**

Weak dependence on discretization of predicted bubble departure diameter and time was observed, as indicated in Figure 16a. The heat flux to the liquid was averaged along a line probe extending radially outwards from the nucleation site and integrated through time starting from the instant of full microlayer depletion. This quantity as well was found weakly dependent on discretization, as indicated in Figure 16b. Results obtained with the finest mesh, of minimum fluid cell size equal to 31.25 microns, are presented in the rest of the paper.
Time step values were chosen in order to respect simultaneously the capillary constraint [32] and the constraint deriving from the Courant-Friedrichs-Levy (CFL) [33] condition. As regards the capillary constraint for a fixed mesh, the maximum admissible time step value is constant throughout the simulation and has been computed following Ref. [29] for the case of $\rho_l \gg \rho_g$ (as is the case for a water–steam system) as $\delta t_{\text{max}} = S \sqrt{\frac{0.5 \rho_l \delta x^3}{\pi \sigma}} = 1.56 \times 10^{-6} \text{s}$, where $S$ is a safety coefficient set equal to 0.25 and $\delta x = 31.25$ micron for the finest grid. At the same time, the time step was checked, and if necessary dynamically adjusted during the simulation [27], its value recalculated at every time step not to exceed a maximum Courant number of 0.25 throughout the computational domain. It was found \textit{a posteriori} that, due to the low flow velocities typical of the pool boiling phenomenon, characterised in conditions of interest by values of the velocity magnitude around 1 m/s, the limiting factor determining the time step was the capillary constraint, not the CFL condition.

![Bubble departure diameter](image)

**Figure 16**

(a) Variation with mesh refinement of the time history of the bubble equivalent diameter; (b) variation with mesh resolution of the cumulative heat flow to the liquid during and after bubble departure; (c) example of a vertical cross-section of the computational domain in the region close to the departing bubble, showing typical discretization (the liquid-vapour interface is colored in red, the solid-fluid interface is represented by a black line).

Sensitivity of predicted bubble behaviour to the input contact angle value has been checked, in order to select a value of the contact angle to match the measured bubble departure time of 20 ms. As indicated in Figure 17, good agreement between measured and modelled bubble departure time was obtained with a contact angle of 40°, corresponding to a predicted bubble departure time of 19.8 ms. This choice of the contact angle is consistent with previous works
[34, 35], where it was recommended a value of the contact angle of $40^\circ$ for boiling of water in atmospheric pressure conditions (101325 Pa and 100°C).

Figure 17
Assessing sensitivity of predicted bubble departure times to contact angle values. In the rest of the paper, results are presented for the $\theta = 40^\circ$ case.

6.2 Qualitative trends in modelled bubble behaviour

Before a comparison of modelled and measured heat flux distribution is presented, it is helpful to illustrate some qualitative trends of modelled bubble behaviour.

Figure 18 shows side views of modelled steam bubble shape at a sequence of times, with the corresponding phase distribution at the solid-fluid interface (the bubble base) from simulation and from experiment. In the sequence of images from numerical simulation, the red ring-shaped region indicates the portion of wall area in contact with the microlayer during the early stages of bubble growth, as captured by the subgrid model described in the preceding text. From about 15 ms onwards, the microlayer has depleted, and portion of wall area enclosed by the three-phase contact line, shown as white circles in the figure, is in contact with vapour. With the choice of numerical settings specified in the preceding text, it was possible to match the simulated microlayer depletion time and bubble lift-off time with the values measured experimentally. The time interval between the two events is here taken as the bubble departure phase. In the simulation, it was possible to match the microlayer depletion time with its experimentally observed value by using the correlation of ref. [31] for the initial thickness of the liquid layer as formed on the solid surface. This correlation has been derived for the case of water in conditions similar to the test cases presented in the current paper. It is understood that other fluids in different conditions of temperature and pressure might exhibit radically different microlayer behavior [36]. The implications on boiling heat transfer might be far reaching, however they are not relevant for the present study, which, as the ref. [31] work, focusses on experimental data collected for water at atmospheric pressure.
Illustrating predicted bubble behaviour. The top sequence of images shows side views of the bubble from interface-capturing simulation. The middle sequence of images shows modelled phase distribution at the solid-fluid interface. An annular region, shown in red, is thermally coupled with the subgrid-scale microlayer depletion model. After microlayer full depletion (occurring at approximately 15 ms into the simulation), the model is deactivated, and the wall is in contact with the actual vapour and liquid phases as predicted by the fluid problem solution. At times after microlayer depletion, the white circle represents the vapour-liquid-solid triple contact line. The bottom sequence of images shows observations of the phase distribution at the solid-fluid interface.
6.2.1 Modelled and measured bubble formation

Comparison between modelled and measured time histories of the bubble equivalent diameter is shown in Figure 19. The comparison indicates a discrepancy between modelled and measured bubble formation.

During the early stages of bubble formation, bubble growth is driven by mass transfer due to evaporation of at the curved surface of the bubble, and from the microlayer beneath the bubble. The observed discrepancy is likely due to differences between the modelled evaporation rates (from the bubble curved surface and from the microlayer) and the actual evaporation rates. However, no direct measurements of the actual evaporation rates are available in conditions of interest, and therefore it has not been possible to check the modelled evaporation rates. Furthermore, the details of the temperature distribution in close proximity to the curved surface of the bubble are not available from experiment (and to the best of our knowledge have not been measured yet in any conditions at all), which again introduces a source of uncertainty.

As regards the later stages of bubble formation, the modelled bubble departure time, in close agreement with measurement, was obtained via tuning of the equilibrium contact angle value imposed in the wall adhesion model. As indicated in Section 6.1, the optimum value of the contact angle was found to be equal to 40°. For present purposes, this choice was found adequate in serving the purpose of matching modelled and measured departure times. However, it is acknowledged that the contact angle value, and more generally the details of the wall adhesion model [37-39], determine to some degree bubble behaviour during the departure phase. The topic is beyond of the scope of the present work and its investigation is deferred to a future study.

![Measured and modelled bubble growth](image_url)

Figure 19
Comparison between modelled and measured time histories of the bubble equivalent diameter, showing agreement between predicted and measured lift-off times.
6.3 Extraction of the heat flux at the solid-fluid interface

During and after bubble departure, i.e. from the instant of microlayer depletion until the eventual bubble lift-off (that is, while the solid surface is exposed to vapour at the bubble base), extraction of heat flow rates at the heat transfer surface, in the direction normal to the solid-fluid interface, is enabled by solution of the conjugate heat transfer problem. The heat flux distribution was found to be essentially rotationally symmetric on the plane of the heat transfer surface and was monitored along a line probe. As indicated in Figure 20a, the line probe extends radially outwards from the centre of the bubble base and passes through the portion of wall area exposed to vapour and then extends some distance into the liquid. The spatially and temporally varying heat flux is computed at each time step from the temperature distribution in the solid as

\[ q'' = k_S \left( \frac{T_S - T_{SF}}{dz/2} \right) \] (1)

the meaning of symbols as indicated Figure 20b, where \( k \) is the thermal conductivity, and the subscripts \( S \), \( SF \) and \( F \) indicate, respectively, the solid, the solid-fluid interface and the fluid. The fluid thermal conductivity takes the appropriate values of the vapour conductivity when the line probe is within the bubble base, and of the liquid conductivity when the line probe extends out into the liquid outside the bubble base. The solid-liquid interface temperature is computed from continuity of the surface-normal heat flux and is equal to:

\[ T_{SF} = \frac{dz k_S T_S + dz k_F T_F}{dz k_F + dz k_S} \] (2)

Figure 20
Extraction of the heat flux component normal to the solid-liquid interface: line probe for heat flux extraction (a), definition of temperature at solid-fluid interface (b).

6.4 Area of influence of enhanced heat transfer mechanism

Matching of predicted and measured microlayer depletion and bubble departure times enabled monitoring the simulated temporal and spatial variation of heat flux at the solid surface during a time interval of duration corresponding to the experimentally observed bubble departure phase. One of the aspects that it was possible to confirm experimentally, thanks to the current set of measurements, concerns the extension of the portion of wall area that is affected by heat transfer augmentation caused by bubble release. This could be verified via consideration of the radial heat flux distribution at a sequence of times, shown in Figure 21.
Using the same approach adopted for the above experimental analysis, the heat flux values have been expressed relative to the baseline heat flux value corresponding to the region unaffected by the presence of the bubble. The line corresponding to the time of 19 ms, approximately 0.8 ms before bubble lift-off, indicates very low values of the heat flux in the dry patch at a short radial distance from the nucleation site. Moving radially outwards, the peak in heat flux occurs in the vicinity of the location of the rewetting front, then it gradually decreases. At later times after bubble lift-off (in the figure, from 21 ms onwards), decay with time of the peak heat flux value is observed. The experiment suggested that the approximately circular area of influence does not extend, in the direction going radially outwards from the nucleation site, for more than about half the bubble departure radius. The same behaviour was observed in the simulation, as shown in Figure 21, indicating that the present simulation methodology could capture experimental trends reliably. The predicted dip in radial heat flux distribution at the periphery of the area of influence was also observed in the experiment, and is due to the bringing of superheated fluid in contact with the wall, as indicated by the temperature distribution shown in Figure 22, causing the local heat flux to decrease to values below the baseline heat flux typical of the region unaffected by the bubble.

Figure 21
Visualization of predicted area of influence of heat transfer augmentation beneath a departing bubble.
Close-up of the qualitative temperature distribution near the heat transfer surface at 1.2 ms after bubble lift off. The white line represents the liquid-vapour interface, the black line indicates the solid-fluid interface. Between approximately 0.5 and 1 mm of distance from the nucleation site, the liquid in the vicinity of the wall is at higher temperature than the wall itself, causing the depression in heat flux values observed in Figure 21 and confirmed by the current experiments (see the following Figure 24 and Figure 25 for a direct comparison of heat flux distributions).

7 COMPARISON BETWEEN MODELLED AND MEASURED HEAT TRANSFER AT BUBBLE DEPARTURE

Directly measured and simulated bubble shapes are compared in Figure 23 at a sequence of times during bubble departure, i.e. the ‘rewet’ phase. In the figure, bubble dimensions have been normalised by the equivalent bubble departure radius, using values of the departure radius from simulation and experiment for displaying, respectively, the modelled (Figure 23a) and measured (Figure 23b) bubble shapes. The simulation was found to predict the gradual decrease in bubble base radius in good agreement with the experiment.
Modelled (a) and measured (b) bubble shape at a sequence of times during the departure process. The x and y axes have been normalized, respectively, by the modelled equivalent departure radius of 1.85 mm for displaying simulation results and by the value of measured equivalent departure radius of 2.5 mm for displaying experimental results. During this phase of bubble behaviour, the bubble base is exposed to vapour and its radial extent reduces as liquid advances on to rewet the solid surface.

For a direct comparison between measured and modelled radial profiles of heat flux at the solid surface beneath the bubble, the radial distance from the nucleation site, i.e. the horizontal axis in Figure 24, has been normalized by the departure radius. In the figure, radial coordinates of the experimental points have been normalized by the measured bubble departure radius, while the modelled bubble departure radius has been used to normalize the heat flux profiles from numerical simulation.

It is noted that some discrepancy is present between modelled and measured bubble dimensions, as discussed in Section 6.2.1. However, the focus of the present work is on bubble departure, and the attendant heat transfer to the liquid, while details of the bubble formation process are of secondary importance in the current investigation. The above normalisation of the radial coordinate has been introduced in order to remove the effect, on the displayed radial heat flux profiles, of discrepancies between modelled and measured bubble formation observed in Section 6.2.1. Thanks to the normalisation, it is possible to observe that both simulation and experiment are in agreement in that both indicate that the heat transfer signature of the departing bubble extends on a portion of the solid surface corresponding to a circle of radius equal to half the bubble departure radius.

There is also good agreement, in terms of the absolute values of the heat flux, between measurement and simulation, with the exception of an initial overestimate of the simulated heat flux peak during the early stages of rewet. Good agreement was observed also of the temporal variation of post-departure heat flux distribution, as indicated by the sequence of profiles in Figure 25, showing the decay in heat flux values typical of the post-departure quench period.
Time sequence of the radial variation of surface-normal heat flux during bubble departure, at the same instants of time of Figure 23. The red lines indicate simulation results, the black squares indicate measurements, the error bars correspond to $\pm 30 \text{ kW/m}^2$. Distance from the nucleation site has been normalized dividing by the value of departure radius. Values of the radial coordinate for the experimental points have been normalized by the measured departure radius, of 2.5 mm. For simulation results, the value of predicted departure radius, of 1.85 mm, has been used.
7.1 Comparison with the ‘Han and Griffith’ [5] transient conduction model

7.1.1 Summary of basic model of heat transfer at bubble departure

Practical CFD simulation of boiling flows, broadly based on the interpenetrating continua (‘two-fluid model’) approach [2], relies on simplified representations of the heat transfer modes associated to the formation and departure of steam bubbles at the solid surface. In the approach [4] used by all industrial CFD methods, three heat transfer modes are postulated and treated separately. These are i) the latent heat transfer associated to the formation of a steam bubble, ii) the heat transfer due to the bringing of cold liquid water in the vicinity of the wall attendant upon bubble departure and iii) the ordinary single phase convection (natural or due to forced flow, respectively for the case of pool or flow boiling) heat transfer in the region unaffected by the bubbles.
Basic model of heat transfer at bubble departure, used as basis of wall-boiling treatments for two-fluid CFD simulation: assumed heat transfer modes (a), ‘quench’ postulated sub-model (from Han and Griffith [5]) (b), indicative axially symmetric heat flux distribution (c) due to transient conduction through the liquid after bubble lift-off.

The above assumed heat transfer modes are sketched in the diagram of Figure 26a, which has been adapted from ref. [40], to which we address the reader for further details of the approach. We are here concerned with the ‘quench’ sub-model sketched in Figure 26b. Its treatment is based on a ‘transient conduction’ physical model due to Han and Griffith [5]. The quench sub-model postulates that at the instant of bubble lift-off cold liquid is suddenly brought into contact with the heated wall from a region at a remote distance from the solid surface. Heat is assumed to be conducted from the wall to the remote liquid. This is modelled as one-dimensional transient conduction of heat through a body of stationary water. The solid surface is assumed at a constant temperature and the body of water is initially assumed at uniform temperature representative of the cold fluid found at a distance from the wall. It is also postulated that the transient conduction process occurs over an area of influence on the solid surface corresponding to a circle of diameter twice as large as the bubble departure diameter, outside of which ordinary convection, unaffected by the presence of the bubble, occurs. This physical situation is sketched in Figure 26c. The spatially uniform heat flux in the area influenced by the bubble varies with time, according to a well-known analytical solution to the problem of one-dimensional transient conduction through a semi-infinite medium, as 

$$q'' = \frac{T_w - T_{\infty}}{\frac{1}{k_l} + \frac{1}{\rho_l c_l}}$$

is constant and uniform. Here $T_w$ is the constant wall temperature, $T_{\infty}$ is a suitable value representative of the remote cold liquid temperature, $k_l$ is the liquid thermal conductivity, $\rho_l$ its density and $c_l$ its specific heat capacity. In real boiling conditions, it is debatable what value ought to be assigned to the remote liquid temperature. For practical modelling purposes, this is normally taken as a suitable constant and uniform ‘bulk’ temperature value, for example the saturation temperature for saturated boiling conditions. The Han and Griffith submodel can be equally applied to both pool [13] [10] and flow boiling conditions [41] [12]. The current paper...
is concerned with saturated pool boiling conditions, for which it has been possible to obtain detailed measurements of the temperature at the solid surface in the vicinity of a departing bubble; flow boiling conditions are outside the scope of the present work.

7.1.2 Checking predictions of the ‘Han and Griffith’ model using measured and interface-capturing-generated results

Measured and simulated heat flux distributions immediately after bubble-lift-off were finally compared with predictions of the Han-Griffith model. In applying the model, the wall temperature is taken as the incipient nucleation temperature of 110.7°C and the remote temperature is taken equal to the saturation temperature of 100°C. Figure 27 indicates that heat flux values returned by the transient conduction model are significantly larger than the measured and interface-capturing modelled heat flux. Likewise, the actual area of influence (from measurement and interface-capturing simulation) is only a fraction of that assumed by the Han and Griffith model. The basic assumption of the Han and Griffith model, that the area of influence of bubble-induced heat transfer augmentation extends for a circle corresponding to two bubble departure diameters, is also at the basis of other popular wall-boiling models [25] [4], widely used in component-scale CFD simulation of boiling phenomena [10]. Results presented here suggest that a reformulation of the models, to include a more accurate estimate of the area of influence, should be incorporated in the development of future wall-boiling treatments.

Figure 27
Comparison between measured radial heat flux distribution immediately after bubble lift-off
and modelled heat flux distribution. Squares indicate experimental values, solid lines correspond to results of interface-capturing simulation and dashed lines indicate predictions of the Han and Griffith model. Different line colors indicate different times. As for the previous figures, heat flux values are expressed relative to the background heat flux value.

8 CONCLUSIONS

This paper presented analysis of the transient cooling of a solid surface beneath a departing steam bubble in pool boiling of water. Sequential measurements of the solid surface temperature distribution have been used to extract the heat flux to the fluid during bubble departure. In conditions of the experiment, the process of bubble formation and lift-off from a surface has been modelled with interface capturing numerical simulation. Key findings of the present work can be summarized as follows:

- The accuracy of the procedure to extract the heat flux to the liquid from a solid surface beneath a departing steam bubble has been checked. The method’s precision enabled a quantitative assessment of the magnitude and spatio-temporal distribution of the heat flux to the fluid during bubble departure and following bubble lift-off.
- The experiment allowed gaining unprecedented insight on the nature of the heat transfer mode associated to the departure of a steam bubble from a solid surface. Heat transfer augmentation was observed taking place during bubble departure, due to rewetting of a portion of wall area exposed to vapour.
- The measurements enabled direct observation of the area of influence of the above heat transfer augmentation mechanism, and clearly indicated that this approximately circular area of influence extends, radially outwards from a nucleation site, for no more than half the bubble departure radius.
- Interface capturing numerical simulation was applied to model bubble formation and release in conditions of the experiment, and enabled extracting the heat flux to the fluid at the solid surface during bubble departure. Good agreement was observed between model and measurement during and after bubble lift-off in terms of the heat flux magnitude, of its spatial distribution on the solid surface, and of its temporal variation. Comparison with measurements confirmed that the interface capturing approach can simulate the process of bubble departure with a good level of accuracy.
- The interface-capturing representation of bubble behaviour requires selecting input parameters of the computational model, namely the bubble contact angle used in the wall adhesion model, and the ‘initial’ microlayer thickness used by the subgrid scale microlayer model. These parameters are known to influence the bubble formation and departure process. For the present work, it sufficed to adjust the contact angle value, and to choose the initial microlayer thickness value predicted by an extant correlation, in order to cause the simulated bubble to depart at a size similar to that observed in the experiment, after a time similar to the measured departure time. It was so possible to analyse the flows of heat to the liquid immediately before and after lift-off in conditions representative of the experiments. However, more effort, outside the scope of this work, is needed in order to
improve our understanding of the effect of bubble contact angle and microlayer thickness values on bubble-growth simulations.

- Predictions of the spatial and temporal variation of the heat flux at bubble departure by a simplified transient conduction model, commonly used as the basis of macroscale CFD models of boiling flows, have been assessed and compared to predictions of the current interface capturing simulation methodology. Experimental data collected during the current work enabled checking the accuracy of the above representations of the heat transfer at bubble departure. Comparison with measurements of the heat flux distribution at bubble departure from interface-capturing simulation and from the simplified transient conduction model indicated that the latter can be in considerable error, at least under present conditions.

9 ACKNOWLEDGEMENTS

The experimental part of this work was supported by the National Research Foundation of Korea (NRF) by the Korea government (MSIT: Ministry of Science and ICT) (No. 2019M2D2A1A02059364) and by “Human Resources Program in Energy Technology” of the Korea Institute of Energy Technology Evaluation and Planning (KETEP), granted financial resource from the Ministry of Trade, Industry & Energy, Republic of Korea (No. 20184030202170).

The PSI-BOIL code is open source software developed by Dr Bojan Niceno and Dr Yohei Sato of Paul Scherrer Institut, available at https://github.com/PSI-NES-LSM-CFD/PSI-Boil.

10 REFERENCES


