Numerical investigation on the nucleate pool boiling heat transfer of R134a outside the plain tube

Shuai-Feng Mao, Wen-Tao Ji, Guo-Hun Chong, Chuang-Yao Zhao, Hu Zhang & Wen-Quan Tao


To link to this article: https://doi.org/10.1080/10407782.2019.1673085

Published online: 17 Oct 2019.

Submit your article to this journal

Article views: 94

View related articles

View Crossmark data
Numerical investigation on the nucleate pool boiling heat transfer of R134a outside the plain tube

Shuai-Feng Mao, Wen-Tao Ji, Guo-Hun Chong, Chuang-Yao Zhao, Hu Zhang, and Wen-Quan Tao

Key Laboratory of Thermo-Fluid Science and Engineering of MOE, Xi’an Jiaotong University, P.R. China

ABSTRACT
In the present work, a numerical method is utilized to simulate the nucleate pool boiling of R134a outside the horizontal plain tube. Eulerian multiphase model integrated with Rensselaer Polytechnic Institute (RPI) nucleate boiling model is adopted. The nucleate pool boiling is numerically investigated at the heat flux of 20–80 kW/m² and saturation temperatures of 279 and 299.7 K. By analyzing the experimental results, a modified correlation of active nucleation site density is proposed. The numerical results based on the modified correlation agree well with theexperimental data of the pool boiling heat transfer outside the plain tube. According to the simulation of azimuthal variations of temperature around the tube, it is found that the region between 90° and 130° has the best heat transfer performance for its high velocity and turbulence. The maximum temperature appears at the upper of the tube because the velocity and turbulence intensity at this region are lower. The investigation helps to explain the phenomena and observed facts in the experiment.

1. Introduction
Nucleate boiling is a basic mode of heat transfer in chemical process, refrigeration industries and nuclear reactors. It has been intensely studied during the last 80 years. For nuclear reactors, it plays an important role in passive decay heat removal system. In the flooded refrigerant evaporators, refrigerant boils outside the tube to cool down the working fluid in the tube. Nucleate boiling heat transfer is also an efficient cooling technique in the thermal management of electrical components [1].

To date, experimental methods continue to play an important role in the investigation of nucleate boiling heat transfer of refrigerants. Many techniques are now available for improvement of boiling heat transfer. In the experiment of Webb and Pais [2], nucleate pool boiling of five refrigerants on five different horizontal tube geometries was investigated. Experimental results show that Thermoexcel-E tube has the highest heat transfer coefficients among these five enhanced tubes. Besides, the variations of wall superheat with angular location for three different tubes were presented to investigate the local heat transfer coefficient. According to the experimental data, GEWA K26 tube has a higher heat transfer coefficient at the bottom than at the top. While GEWA SE tube has the opposite heat transfer distribution and Turbo-B tube has the
constant heat transfer coefficient. By testing a wide variety of tubes, it provides the guidelines for the design of enhanced geometries and high performance heat exchangers.

Base on the experimental data, several correlations were proposed to predict the boiling heat transfer coefficient. For refrigerant, Cooper equation [3] is often used to predict heat transfer coefficient and test the reliability of experimental data for boiling heat transfer outside the plain surface. Rooyen and Thome [4] tested the pool boiling heat transfer coefficient of two enhanced tubes with R-134a, R-236fa and R-1234ze(E). Based on their experimental data, an enhanced tube pool boiling correlation was proposed. However, due to the complexity of boiling heat transfer, the predicted results deviated from the real values by more than 100% [5].

With the development of computational fluid dynamics (CFD), numerical simulations of boiling have received extensive attentions. Perez-Raya and Kandlikar [6] proposed a simulation method for nucleate boiling which considers mass transfer with interfacial temperature gradients and a sharp interface. Simulation results at 6.2K wall superheat using this method show a favorable agreement with experimental data.

Sato and Niceno [7] used an interface tracking method to simulate the pool boiling from nucleate to film boiling regime. Their method only has one empirical parameter that takes the micro-layer into consideration using a specialized model. The computed boiling heat transfer coefficient fitted well with the experimental data. Sadeghi et al. [8] performed a numerical investigation on the film boiling around the elliptical tubes. They proposed a novel correlation accounting for the elliptical tube’s diameter and aspect ratio. Saturated pool film boiling over a flat
horizontal surface was investigated numerically by Ningegowda and Premachandran [9]. Single-mode and multimode film boiling model were used in the numerical simulation to understand the characteristics of flow. They found that the average Nusselt number calculated by multimode is lower than that in single-mode.

Direct numerical simulation of nucleate pool boiling at large microscopic contact angle and moderate Jakob number was presented by Huber et al. [10]. They found that the influence of the wall superheat on the contact angle and contact line heat flux is weak. A simplified correlation on the dimensionless bubble detachment variation with the Jakob number was proposed after the simulation.

Recently, it was found that the Eulerian multiphase model with the Rensselaer Polytechnic Institute model (RPI) [11] can simulate the pool boiling fluid flow successfully [12–16]. Zhang et al. [15] performed a CFD simulation on nucleate pool boiling of liquid nitrogen using the RPI model. The numerical results also fitted quite well with the experimental data. They compared the bubble departure diameter calculated from existing correlations with the measured values and found that the existing correlations under-predict the bubble departure diameter.

Minocha et al. [12] performed a 3D CFD simulation of passive decay heat removal system under boiling conditions. They investigated the effect of inclination of tube on sliding bubble dynamics and heat transfer. It was found that the optimum angle is 75°. Similar work was performed by Noori et al. [14]. They performed numerical investigation into the inclination effect on conjugate pool boiling in a passive heat removal system. They also took the condensation of steam in the tube into consideration. The results show that the optimum angle for heat transfer is between −60° and −30°. Additionally, nucleate boiling on staggered tube bundles was also investigated by 2D CFD simulation [13]. The effect of saturation temperature, the variation in the heat transfer and the effect of working fluid were investigated numerically. The success of applying RPI model to numerical simulation of pool boiling help us predict the pool boiling behavior and heat transfer accurately and design advanced nuclear reactors [12, 17].

Although there are many experimental investigations on the boiling heat transfer of refrigerant outside the tubes, numerical investigations on pool boiling are still quite few to the author’s knowledge. Normally, it is hard to observe the dynamic process of boiling with the experimental method and perform experiments at the extremely low temperature. It is worthy to undertake the research on nucleate pool boiling heat transfer with the numerical approaches, for its important role in engineering practice. In order to numerically investigate the pool boiling and predict the boiling heat transfer coefficient accurately, Eulerian multiphase model with RPI model is utilized in this work.

The rest sections are arranged as follows: firstly, the descriptions of mathematical formulation and numerical method are presented. Secondly, comparison with experimental results is presented and a modified correlation of active nucleation site density is proposed. Then, the obtained numerical results including effect of interfacial force and circumferential variation of boiling heat transfer are discussed. Finally, the work is concluded.

2. Mathematical formulation

2.1. Governing equations

In the present numerical study, Eulerian multiphase model is adopted to simulate pool boiling heat transfer outside the plain tubes. In Eulerian multiphase model, mass, momentum, energy conservation equations are solved for each phase:
Mass equation:

$$\frac{\partial}{\partial t}(\alpha_i \rho_i) + \nabla \cdot (\alpha_i \rho_i \vec{u}_i) = \sum_{j=1}^{n} (\dot{m}_{ji} - \dot{m}_{ij}) + S_i$$  (1)

Momentum equation:

$$\frac{\partial}{\partial t}(\alpha_i \rho_i \vec{u}_i) + \nabla \cdot (\alpha_i \rho_i \vec{u}_i \vec{u}_i) = -\alpha_i \nabla \rho + \nabla \cdot \vec{P}_i + \alpha_i \rho_i \vec{g} + \sum_{j=1}^{n} (\dot{m}_{ji} \vec{u}_j - \dot{m}_{ij} \vec{u}_i) + \vec{S}_{u,i}$$  (2)

where $\vec{P}_i$ is the $i$-th phase stress-strain tensor:

$$\vec{P}_i = \alpha_i \mu_i (\nabla \vec{u}_i + (\nabla \vec{u}_i)^T) + \alpha_i \left( \sqrt{\frac{2}{3}} \mu_i \right) \nabla \cdot \vec{u}_i \vec{l}$$  (3)

Energy equation:

$$\frac{\partial}{\partial t}(\alpha_i \rho_i h_i) + \nabla \cdot (\alpha_i \rho_i \vec{u}_i h_i) = \alpha_i \frac{\partial P}{\partial t} + \vec{P}_i : \nabla \vec{u}_i - \nabla \cdot \vec{q}_i + \sum_{j=1}^{n} (Q_{ji} + \dot{m}_{ji} h_{ji} - \dot{m}_{ij} h_{ij}) + S_{h,i}$$  (4)

where subscripts “i” and “j” denote the $i$-th phase and $j$-th phase. $\alpha_i$, $\vec{u}_i$, $\rho_i$, $P$, $\vec{P}_i$, $h_i$ and $\vec{q}_i$ denote volume of fraction, velocity, density, pressure, stress tensor, specific enthalpy and heat flux for $i$-th phase respectively. Additionally, $\dot{m}_{ji}$ characterizes mass transfer from $j$-th phase to $i$-th phase, $Q_{ji}$ is the intensity of heat exchange between phases, $h_{ji}$ is the interphase enthalpy and $S_i$, $S_{u,i}$, $S_{h,i}$ are mass, momentum and energy source term.

There are three methods for modeling turbulence in multiphase flow with the $k-\varepsilon$ and $k-\omega$ models. The three methods are Mixture, Dispersed and Per Phase methods. In mixture turbulence model, different phases are treated as a mixture and mixture properties and velocities are used in one set of turbulence equations. For the Dispersed method, liquid phase is treated as continuous phase and turbulent equations are solved for the continuous phase. Then theory [18] is used to predict turbulent quantities for the dispersed phases. For the Per Phase method, liquid and vapor phases are treated as two independent phases and turbulence equations are solved for each phase. As a result, this method will take more computing time.

In this work, the two-equation mixture turbulence model is adopted. The descriptions based on the realizable $k-\varepsilon$ are shown as follows:

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla \cdot (\rho_m \vec{u}_m k) = \nabla \cdot \left[ \left( \mu_{L,m} + \frac{\mu_{t,m}}{\sigma_k} \right) \nabla k \right] + \left[ \mu_{t,m} \left( \nabla \vec{u}_m + (\nabla \vec{u}_m)^T \right) : \nabla \vec{u}_m \right] - \rho_m \varepsilon + S_{km}$$  (5)

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla \cdot (\rho_m \vec{u}_m \varepsilon) = \nabla \cdot \left[ \left( \mu_{L,m} + \frac{\mu_{t,m}}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} \left( C_1 \varepsilon \left[ \mu_{t,m} \left( \nabla \vec{u}_m + (\nabla \vec{u}_m)^T \right) : \nabla \vec{u}_m \right] \right) - C_2 \rho_m \varepsilon + C_3 \varepsilon \frac{\varepsilon}{k} G_b + S_{\rho_m}$$  (6)

where

$$C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \eta = \frac{k}{\varepsilon}, S = \sqrt{2S_1 S_{ij}}$$  (7)

The eddy viscosity is defined as:

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon}$$  (8)

where $k$ and $\varepsilon$ denote the turbulent kinetic energy and dissipation rate respectively. Different
from other $k$-$\epsilon$ model, $C_\mu$ is no longer constant and computed from:

$$C_\mu = \frac{1}{A_0 + A_S k U^*} \quad U^* = \sqrt{S_{ij}S_{ij} + \Omega j \Omega j}, \quad \Omega j = \Omega j - 2 \varepsilon_{ijk} \omega_k, \quad \Omega j = \Omega j - \varepsilon_{ijk} \omega_k$$

where $\Omega j$ denotes the mean rate of rotation tensor in a moving reference frame with the angular velocity $\omega_k$. The other parameters and constants are as follows:

$$A_0 = 4.04, A_S = \sqrt{6} \cos \phi$$

$$\phi = \frac{1}{3} \cos^{-1}(\sqrt{W}), \quad W = \frac{S_{ijk}S_{ijk}}{S^2}, \quad S = \sqrt{S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$$

$$C_{2e} = 1.92, C_{3e} = \tan \left( \frac{\nu}{U} \right), \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3$$

In the equations above, the mixture density, $\rho_m$, molecular viscosity, $\mu_m$, and velocity, $\mathbf{u}_m$, are computed from:

$$\rho_m = \sum_{i=1}^{n} x_i \rho_i$$

$$\mu_m = \sum_{i=1}^{n} x_i \mu_i$$

$$\mathbf{u}_m = \frac{\sum_{i=1}^{n} x_i \rho_i \mathbf{u}_i}{\sum_{i=1}^{n} x_i \rho_i}$$

The mixture turbulent viscosity $\mu_{t,m}$, and production of turbulence kinetic energy, $G_{k,m}$, are computed from:

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\epsilon}; \quad G_{k,m} = \mu_{t,m} (\nabla \mathbf{u}_m + (\nabla \mathbf{u}_m)^T) : \nabla \mathbf{u}_m$$

The turbulent viscosity for $i$-th phase is computed from:

$$\mu_{t,i} = \frac{\rho_i}{\rho_m} \mu_{t,m}$$

Turbulence in boiling flow is extremely complex, because the breakup, coalescence and interaction of the bubbles can also affect the turbulent flow. To account for turbulent interaction, the Sato model [19] is used for turbulent viscosity:

$$\nu_q = C_{\mu} \frac{k^2}{\epsilon_q} + C_{\mu,p} \sum p d_p | \mathbf{U}_p - \mathbf{U}_q |$$

### 2.2. Interfacial momentum transfer

The interfacial momentum transfer between liquid and vapor phase consists of the drag force ($F_i,\text{drag}$), lift force ($F_i,\text{lift}$), virtual mass force ($F_i,\text{vm}$), turbulent dispersion force ($F_i,\text{dispersion}$) and wall lubrication force ($F_i,\text{wall}$). These forces are accounted for by the term $\mathbf{S}_{u,i}$:

$$\mathbf{S}_{u,i} = F_i,\text{drag} + F_i,\text{lift} + F_i,\text{vm} + F_i,\text{dispersion} + F_i,\text{wall}$$

The drag force which acts as a hydrodynamic resistance to the motion of the bubbles through the liquid phase is essential for the calculation. The drag law is different for different bubbly flow patterns. So, three different drag correlations are included in the Grace drag model [20] for
different flow regimes. It has been proved that the most appropriate drag which is in good agree-
ment with the measurements is the Grace Drag law [21]:

\[ F_{i,\text{drag}} = \frac{1}{8} C_{D} A_{\text{interfacial}} \rho | \vec{u}_j - \vec{u}_i | (\vec{u}_j - \vec{u}_i) \]

\[ C_D = \max \left[ \min (C_{D, \text{ellipse}}, C_{D, \text{cap}}), C_{D, \text{sphere}} \right] \]

\[ C_{D, \text{sphere}} = \begin{cases} \frac{24}{\text{Re}} & \text{Re} \leq 0.01 \\ \frac{24}{\left( 1 + 0.15 \text{Re}^{0.687} \right)} & \text{Re} > 0.01 \end{cases} ; \quad C_{D, \text{cap}} = \frac{8}{3} \]

\[ C_{D, \text{ellipse}} = \frac{4 \text{g} d_i^2 (\rho_i - \rho_j)}{3 U_i^2} ; \quad U_i = \frac{\mu_i}{\rho_i d_j} \text{Mo}^{-0.149} (J-0.857) ; \]

\[ \text{Mo} = \frac{(\mu_i)^4 g (\rho_i - \rho_j)}{(\rho_j)^3 \sigma^3} \]

\[ J = \begin{cases} 0.94 H^{0.757} & 2 < H \leq 59.3 \\ 3.42 H^{0.441} & H > 59.3 \end{cases} ; \quad H = \frac{4}{3} \text{EoMo}^{-0.149} \left( \frac{\mu_i}{0.0009} \right)^{-0.14} \]

\[ \text{Mo} = \frac{g (\rho_i - \rho_j) (d_i)^2}{\sigma (1 + 0.163 \text{Eo}^{0.757})^{2/3}} \]

where \( C_D \) is the drag force coefficient, \( A_{\text{interfacial}} \) is the interfacial area concentration, \( \text{Re} \) is Reynolds number, \( \text{Mo} \) is Morton number and \( \text{Eo} \) is Eotvos number.

Auton et al. [22] pointed out that when the multiphase flow is non-uniform and rotational, there exists a transverse force called lift force. Including lift force in simulations could be important for inhomogeneous radial distribution of bubble’s holdup. The lift force calculated from the Tomiyama model [23] is suitable for all shape and size of bubble.

\[ F_{i,\text{lift}} = C_L \mu_i \rho_i (\vec{u}_j - \vec{u}_i) \times (\nabla \times \vec{u}_i) \]

\[ C_L = \begin{cases} \min [0.288 \tanh (0.121 \text{Re}) f (\text{Eo}_d)] & \text{Eo} < 4 \\ 0.00105 \text{Eo}_d^3 - 0.0159 \text{Eo}_d^2 - 0.0204 \text{Eo}_d + 0.474 & 4 < \text{Eo} < 10 \\ -0.29 & \text{Eo} > 10 \end{cases} \]

\[ Eo_d = \frac{g (\rho_i - \rho_j) (d_i)^2}{\sigma (1 + 0.163 \text{Eo}^{0.757})^{2/3}} \]

The wall lubrication force that prevents the bubbles from touching the wall is calculated using the Antal model [24].

\[ F_{i,\text{wall}} = C_w \mu_i \rho_i | \vec{u}_j - \vec{u}_i |^2 n_w \]

\[ C_w = \max (0, \frac{C_{w1}}{d_p} + \frac{C_{w2}}{y_w}) \]

where \( C_{w1} = -0.01 \) and \( C_{w2} = 0.05 \) are non-dimensional coefficients; \( d_p \) is the bubble diameter and \( y_w \) is the distance to the nearest wall.

Another essential interfacial force is the turbulent dispersion force. It transports the bubbles from the heating wall to the bulk saturated liquid area. The turbulent dispersion force which arises from averaging the interphase drag term is calculated from Burns model [25].

\[ F_{i,\text{dispersion}} = C_D \mu_j \frac{t}{\rho_j \text{Sc}_i} \left( \nabla \frac{\rho_i}{\rho_j} - \nabla \frac{\rho_i}{\rho_i} \right) \]

where \( \text{Sc} \) is Schmidt number.
The virtual mass force is caused by the inertia of the dispersed phase. It is important for some practical flow such as strongly accelerating flow and multiphase flow with large continuous-dispersed density ratio. Frank et al. [26] pointed out the effect of virtual mass force is relatively minor in importance at steady multiphase flow. The virtual mass force which occurs when the vapor phase accelerates relative to the liquid phase [27] is defined as follows:

\[ F_{i,vm} = -F_{j,vm} = 0.5\alpha_j\rho_j \left( \frac{D\bar{u}_j}{Dt} - \frac{D\bar{u}_i}{Dt} \right) \]  

(23)

2.3. Boiling model

The Rensselaer Polytechnic Institute (RPI) wall boiling model proposed by Kurul and Podowlki [11] is adopted to model pool boiling outside the tube. According to the basic RPI model, the total heat flux coming from the tube wall can be divided into three parts, the convective heat flux \( q_C \), the evaporative heat flux \( q_E \) and the quenching heat flux \( q_Q \).

\[ q_w = q_C + q_E + q_Q \]  

(24)

The convective heat flux is calculated from

\[ q_C = h_C(T_w - T_l)(1 - a_b) \]  

(25)

where \( h_C \) is the convective heat transfer coefficient, \( T_w \) and \( T_l \) are the wall and liquid temperature respectively, and \( a_b \) is the surface covered by the bubbles and is calculated as follows [28]:

\[ a_b = \min \left( 1, K_{DK} \frac{N_W\pi D_W^2}{4} \right); \quad K_{DK} = 4.8 \exp \left( -\frac{1}{80} \frac{\rho_l C_{p,l}(T_{sat} - T_l)}{\rho_s h_l} \right) \]  

(26)

where \( N_w \) is the nucleation site density that is calculated by the correlation of Lemmert and Chawla [29]:

\[ N_W = C^n(T_w - T_{sat})^n; \quad C = 210, \quad n = 1.805 \]  

(27)

The bubble diameter for the RPI model is based on the correlation of Unal [30]:

\[ D_W = \min \left( 0.0014, 0.0006 \exp \left( -\frac{\Delta T_w}{45} \right) \right) \]  

(28)

After bubble detaching from the tube wall periodically, the liquid fills the wall vicinity. The quenching heat flux \( q_Q \) which models this transient energy transfer is expressed as:

\[ q_Q = \frac{k_l}{\sqrt{\pi k_l \rho_l C_{p,l} \tau}} (T_w - T_l) \]  

(29)

where \( k_l \) is the conductivity, \( \tau \) is the periodic time of bubble detachment and is calculated by the correlation of Cole [31]:

\[ \frac{1}{\tau} = \sqrt{\frac{4g(\rho_l - \rho_v)}{3\rho_l D_W}} \]  

(30)

The evaporative heat flux is given by:

\[ q_E = \frac{\pi}{6} D_W^3 N_W \rho_v h_l \]  

(31)
2.4. Heat and mass transfer mechanisms

The place where mass transfer occurs can be divided into two regions: near wall area and bulk saturated liquid area. The evaporation mass transfer near the wall is derived from the evaporation heat flux:

\[
m_{\text{boiling},1} = \frac{q_E}{h_v + C_p,l(T_{\text{sat}} - T_i)}
\]

The mass transfer in the bulk saturated liquid area is calculated as:

\[
m_{\text{boiling},2} = \frac{q_l + q_v}{h_v}
\]

\[
q_l = \frac{k_l Nu}{D_W} (T_{\text{sat}} - T_i),
q_v = \frac{\alpha_v \rho_v C_{p,v}}{\delta t} (T_{\text{sat}} - T_v)
\]

where \( q_l \) is the heat transfer from the bubble to the liquid as the bubble detaches from the wall and moves to the bulk liquid area. The Nusselt number, \( Nu \) is calculated by the Ranz-Marshall model [32]:

\[
Nu_l = 2 + 0.6 R e^{1/2} P r^{1/3}
\]

The \( q_v \) denotes the heat transfer from the interface to vapor. It is defined as:

\[
q_v = \frac{\alpha_v \rho_v C_{p,v}}{\delta t} (T_{\text{sat}} - T_v)
\]

where \( C_{p,v} \) is the isobaric heat capacity and \( \delta t \) is the time scale set to 0.05 [33].

3. Numerical method

3.1. Physical model

The present study focuses on a horizontal plain tube in a refrigerant pool. As depicted in Figure 1, the computational domain consists of a pool and a tube with outer diameter of 19mm. According to Huber et al. [10], the tube thickness can be ignored without affecting the numerical results. In seek of an accurate prediction and reduction of computational costs, structured grid created by ICEM CFD was adopted. As shown in Figure 1, O-grid Block generation technique was used to reduce skewness. It should be noted that the boundary layer of the tube should have a denser mesh. In the present numerical study, a boundary layer with an aspect ratio of 1.2 and 27 rows was used. The value of \( y \) plus around the tube is between 1 and 2.

3.2. Assumptions and solution method

The following assumptions are made for the pool boiling numerical simulation:

1. The thermo-physical properties of liquid and vapor phase are constant under the specified operating condition.
2. The pool boiling flow is turbulent and transient.
3. The contact angle for refrigerant R134a between the liquid and vapor on the tube is constant, 6.5°.
4. The thickness of tube is ignored and constant heat flux is set on the tube [10].
The boundary condition for the tube is constant heat flux. Pressure inlet condition is applied for the upper side of the pool to get the specified operating condition. No slip and adiabatic condition was applied for the pool wall.

Figure 1. Schematic of the calculation domain and the grid system.
The numerical simulation of pool boiling was performed on the commercial CFD software ANSYS Fluent 19.0. The coupled scheme coupled with volume fractions was employed for pressure-velocity coupling. The gradient was calculated by the least squares cell based method and HRIC method was selected for the volume fraction. Second order upwind discretization was selected for the momentum, turbulent kinetic energy, turbulent dissipation rate and energy terms. For transient formulation, first order implicit formulation was used.

For solution initialization, the pool was filled with refrigerant R134a with the corresponding saturation temperature. A surface monitor for the average temperature of the tube was created. Transient simulations continued until a converged tube wall temperature was obtained. Figure 2 shows the temperature change during the simulation. The wall temperature is converged after 5s, and hence the wall temperature at 5s was adopted to calculate the boiling heat transfer coefficient.

4. Results and discussion

4.1. Grid independence

Firstly, the grid independence study is performed with three different numerical grids. As shown in Figure 3, the variation of the heat transfer coefficient with heat flux is depicted for three different cells. The plots in Figure 3 show that the numerical results are independent of the grid. Hence, the medium cell with 12528 cells is adopted for its good compromise between the calculation time and accuracy.

4.2. Time-step independence

The time step is an important parameter for the transient numerical simulation. A too large time step may cause divergence while a too little time step will cost too much calculation time. The effect of time step on the pool boiling curve is depicted in Figure 4. As shown in Figure 4, the numerical results are basically independent using three time steps. The time step 0.001s is used in the simulation.

4.3. Validation and comparison with experimental results

4.3.1. Experimental apparatus and procedures

As shown in Figure 5, the experimental apparatus consists of three circulating loops: refrigerant, heating and cooling water. The refrigerant circulating system has three main parts: the boiler, the
condenser and the ducts connecting the two vessels. When testing the nucleate pool boiling heat transfer coefficient of the tube, the hot water is flowing through the tested tube fixed in the boiling vessel and the refrigerant R134a is boiling outside the test tube. Then the refrigerant which has converted to vapor rises to the upper condenser through the duct. After reaching the

Figure 3. Variation of the pool boiling heat transfer coefficient with grid number.

Figure 4. Variation of the pool boiling heat transfer coefficient with time step.

Figure 5. Schematic diagram of the experimental apparatus: (1) evaporator; (2) condenser; (3) thermocouple; (4) pressure gauge; (5) condensate measuring container; (6) exhausting valve; (7) subsidiary electric heater; (8) weight-time flow meter of cooling water; (9) cooling water pump; (10) cold water storage tank; (11) weight-time flow meter of heating water; (12) heating water pump; (13) hot water storage tank.
condenser, the vapor refrigerant is condensed outside the condensing tube in which cooling water flows and then returns to the boiler by gravity. The hot water is heated by electric heater in the hot water storage tank.

A pressure gauge is used to measure the pressure of the boiler. The test range is 2.5 MPa with precision of ±0.00625 MPa. The flow rate of hot water inside the tested tube is measured by a weight-time flow meter. The temperature of R-134a in different parts of the boiler is measured by five platinum temperature transducers with a precision of ±(0.15 + 0.002|T|) K (T is real temperature). The water temperature difference of the inlet and outlet is measured with two thermocouples.

Before the experiment, tightness check was performed firstly. High pressure nitrogen with a pressure of 1.2 MPa was charged into the system. This pressure should be maintained for at least 24 hours to prevent the leaks. After the tightness check, high pressure nitrogen was discharged and then the system was evacuated to an absolute pressure of at least 800Pa. A small quantity of R134a was charged into the boiler and then evacuate. This process was repeated three times to remove the other gas. Finally, the refrigerant was charged into the system and then the experiment was conducted. Pool boiling experiment was conducted at the saturation temperature of 279 K. The external diameter of the tube is 19 mm, which is the same as that in simulation.

### 4.3.2. Model validation with experimental data

Figure 6(a) shows the comparison between the present numerical results and experimental data. More validation simulations were performed at the saturation temperature of 299.7 K in
comparison with the experimental data of Webb and Pais [2] (Figure 6(b)). As can be seen, the maximum deviation is \(-16\%\).

The deviation is larger at lower and higher heat flux. This deviation might partially be attributed to the empirical parameters in RPI model. The correlations for nucleation site density are mostly based on the experimental data of water [13]. As shown in Figure 6, this correlation based on water over-predicts the heat transfer coefficient at lower heat flux \((q < 30 \text{ kW/m}^2)\) and under-predicts the heat transfer coefficient at higher heat flux \((q > 40 \text{ kW/m}^2)\). Since the difference does exist in thermal properties, the nucleation site density should be different for the refrigerant. The surface tension, viscosity and latent heat of hydro-fluorocarbon (HFC) and hydrofluoro-olefin (HFO) refrigerants are all much less than water. For pool boiling of R134a, a small amount of nucleation site density is observed according to the experimental investigation of the authors [34–36]. In the present simulation, a modified correlation is provided to obtain the nucleation site density and the heat transfer coefficient:

\[
N_W = \begin{cases} 
C^n(T_W - T_{sat})^n; & C = 210, \ n = 1.75, \ q \leq 30 \text{ kW/m}^2 \\
C^n(T_W - T_{sat})^n; & C = 210, \ n = 1.85, \ q > 30 \text{ kW/m}^2
\end{cases}
\]  

(37)

At different stage of heat flux, the empirical parameter \(n\) has different values. Figure 7 shows the comparison between the experimental results and numerical simulation results using the modified correlation. As shown in Figure 7, the deviation is lower than the result for water. At the saturation temperature of 279 K, the deviation reduced from \(-14\%\sim-9\%\) to \(-7\%\sim8\%\).
4.4. Effect of saturation temperature

In Figure 8, the variation of the heat transfer coefficient with the heat flux is illustrated at two different saturation temperatures. It can be observed that the boiling heat transfer coefficient at the higher saturation temperature of 299.7 K is 16% higher than that at 279 K. The boiling heat transfer coefficient at two different saturation temperatures both increases linearly with the increment of heat flux. The constant slope is in good qualitative agreement with the Cooper equation [3].

To further investigate the effect of saturation temperature, the variation of the vapor volume fraction with the heat flux at two different saturation temperatures is depicted in Figure 9. According to Figure 9, the vapor volume fraction at saturation temperature of 279 K is higher than that at 299.7 K. Additionally, the slope of vapor fraction versus heat flux is higher at saturation temperature of 279 K than that at 299.7 K. The simulated results are in consistent with the numerical results of Ahmadpour et al. [13], in which they concluded that the vapor volume fraction increases when the saturation temperature reduces. This could explain the difference of heat transfer coefficient at different saturation temperatures. The high level of the vapor volume fraction at the tube surface means low frequency of bubble departure and larger bubble diameter. It might prevent the saturated liquid refrigerant from contacting with the nucleation sites outside the tube and degrade the heat transfer performance. The results are also compatible with the experimental data of Akiyama et al. [37]. They found that with the decrease of saturation pressure (saturation temperature), formation of bubbles is gradually changing from very small to large.
4.5. The role of interfacial force in the calculation

In order to obtain an accurate representation of boiling, it is necessary to understand the mechanisms of interfacial force effects. For gas-liquid two-phase flow without boiling, the effects of drag, lift, wall lubrication and turbulent dispersion forces have been discussed in Yamoah et al.’s work [38]. Based on the numerical results, a set of Grace drag coefficient model [21], Tomiyama lift coefficient model [23], Antal et al.’s wall force model [24] and Favre averaged turbulent dispersion force [25] were recommended. As discussed previously in Section 2, these forces are particularly important for the pool boiling. However, the role of interfacial force in pool boiling calculation has not been clearly identified yet. Hence, simulations are performed to identify the specific effect of these forces.

As shown in Figure 10, Case 1 is the standard numerical case which includes all the interfacial forces. Case 2 is that without the effect of turbulent interaction. The effect is obvious according to the simulation. It has 7%–28% lower boiling heat transfer coefficients in the region of 50–80 kW/m² compared with the standard case (Case 1). The effect of wall lubrication (Case 3) and lift (Case 4) seems to be less according to the investigation.

The numerical results for vapor volume fraction at the tube surface are displayed in Figure 11. It can be observed that the numerically predicted level of the vapor volume fraction in Case 2 is much higher than other results in the region of 50–80 kW/m². The high level of the vapor volume fraction leads to the lower heat transfer coefficient at the higher heat flux (Figure 10). In the near wall region, the interaction of nucleate bubbles with the turbulent flow can speed up the
departure frequency of bubbles and improve the heat transfer performance. It is important to take the turbulence interaction force into consideration in the simulation.

The numerically predicted level of the vapor volume fraction in Case 3 is higher than that in Case 1 by 13%. Wall lubrication force may help to push bubbles away from the tube surface. The heat transfer coefficient and vapor volume fraction in Case 4 both agree well with those in Case 1. This indicates that the lift force is of minor importance in pool boiling simulations.

4.6. Circumferential variation of boiling heat transfer

The circumferential variation of the temperature exists during the nucleate pool boiling on a horizontal plain tube. It can be affect by: heat flux, pressure, tube diameter, thermo-physical properties and wall material. Dominiczak et al. [39] measured the circumferential temperature distribution and proposed an approach to avoid the circumferential variation of temperature.

The circumferential temperature distribution at different heat flux in present numerical study is shown in Figure 12. The plots in Figure 12 all indicate that the maximum temperature appears at the upper of the tube while the minimum temperature appears at the region between 90° and 130° (as shown in Figure 1(b)). The temperature profile is in good qualitative agreement with direct observation. The largest difference of temperature in the circumferential region is about 0.5 K. As the increase of heat flux, the variation becomes more obvious. As the increase of the heat flux, the variations of turbulence and velocity around the tube are becoming more apparent (Figure 13 and Figure 14). It might be the reason for the larger difference of temperatures around the tube surface at higher heat flux.

![Figure 12. Azimuthal variations of temperature for the tube at saturation temperature of 279 K.](image1)

![Figure 13. Azimuthal variations of liquid velocity at 1mm away from the tube at saturation temperature of 279 K.](image2)
Azimuthal variations of liquid velocity at 1mm away from the tube are illustrated in Figure 13. It can be found that the maximum velocity appears at the region of $90^\circ < \theta < 120^\circ$, where the minimum temperature appears. Similar results are also observed in Ahmadpour et al.’s work [13].

Moreover, maximum turbulent kinetic energy appears at the same region (Figure 14). The high heat transfer performance at that region may be attributed to the combined effect of high velocity and turbulent kinetic energy. In contrast, the upper of the tube has the minimum velocity and turbulent kinetic energy, which results in the lowest heat transfer performance at the region.

Figure 15 shows the simulation result of velocity vectors for R134a at 30 kW/m² and saturation temperature of 279 K.

Azimuthal variations of liquid velocity at 1mm away from the tube are illustrated in Figure 13. It can be found that the maximum velocity appears at the region of $90^\circ < \theta < 120^\circ$, where the minimum temperature appears. Similar results are also observed in Ahmadpour et al.’s work [13]. Moreover, maximum turbulent kinetic energy appears at the same region (Figure 14). The high heat transfer performance at that region may be attributed to the combined effect of high velocity and turbulent kinetic energy. In contrast, the upper of the tube has the minimum velocity and turbulent kinetic energy, which results in the lowest heat transfer performance at the region.

Figure 15 shows the simulation result of velocity vectors for R134a at 30 kW/m² and saturation temperature of 279 K. Figure 16 shows a picture from high-speed video in pool boiling at
23kW/m² and saturation temperature of 279 K. The velocity vectors in simulation are qualita-
tively in consistent with the experimental phenomena as depicted in Figure 16. As can be seen in
the picture, the buoyancy induces the fluid flows along the tube surface. The minimum velocity
exists at the bottom of the tube and the velocity increase along the tube height.

The maximum temperature point in present numerical study is in agreement with the experi-
mental data of Dominiczak et al. [39]. However, according to their experimental data, the min-
uminimum temperature appears at the bottom of the tube. Another work about circumferential
temperature distribution was conducted by Webb and Pais [2]. The three enhanced tubes in their
experiment have three different circumferential temperature distributions. The local heat transfer
coefficient can be affected by local boiling parameters: bubble diameter, frequency and nucleate
site density. These parameters are related to the tube surface which is not considered in the RPI
model. So it limits the application of RPI model on enhanced tubes.

In the present investigation, the pool boiling heat transfer of R134a around the plain tube was
investigated. The simulation result agrees well with the experimental data. Overall, the RPI model
shows promises in predicting the nucleate pool boiling heat transfer coefficient outside the tube.
However, for the enhanced tubes, the prediction result is inaccurate. The investigation in this
paper helps to explain the phenomena and observed facts in the experiment. Especially the distri-
bution of velocity and turbulent intensity around the tube, it is difficult to measure in the prac-
tical experiment.

5. Conclusions

In this study, numerical simulation was performed for pool boiling of R134a outside the horizon-
tal single plain tube. An Eulerian multiphase model coupled with the RPI model is used to pre-
dict the heat transfer coefficient. According to the results, the following conclusions can
be drawn:

1. The numerical results agree well with the experimental data for the pool boiling heat transfer
outside the plain tube. The numerical method using Eulerian multiphase model with the RPI
model can predict the pool boiling heat transfer coefficient of R134a accurately. For pool
boiling simulations, it is important to take the turbulent interaction into consideration. The
wall lubrication force can reduce the vapor volume fraction. The lift force has minor effect
on the pool boiling heat transfer simulation.

2. Based on the present numerical results, a modified correlation based on Lemmert and
Chawla’s is provided. By tuning the coefficient $n$ for refrigerant at different stage of heat
flux, a more accurate boiling heat transfer coefficient is obtained.
3. The vapor volume fraction at the tube tends to increase with saturation temperature. Consequently, the heat transfer coefficient increases with the increase of saturation temperature.

4. The region between 90°C and 130°C has the best heat transfer performance for its high velocity and turbulence. The maximum temperature appears at the upper of the tube because the velocity and turbulence at this region are lower.

**Funding**

This work was supported by the National Natural Science Foundation of China (51776160) and National Key R&D Program of China (No. 2016YFB0601200).

**Reference**


