Journal of the Serbian Society for Computational Mechanics / Vol. 15 / No. 1, 2021 / pp 177-185
(10.24874/jsscm.2021.15.01.12)

HEAT TRANSFER TO A BOILING LIQUID – NUMERICAL STUDY

Andrijana Stojanović*, Srđan Belošević, Nenad Crnomarković, Ivan Tomanović, Aleksandar Milićević

Department of Thermal Engineering and Energy, VINČA Institute of Nuclear Sciences - National Institute of the Republic of Serbia, University of Belgrade, Belgrade, Serbia
e-mail: andrijana@vin.bg.ac.rs
** corresponding author*

Abstract

Due to extensive research efforts within the past thirty years, the mechanisms by which bubbles transfer energy during pool boiling are relatively well understood and have various applications in reactors, rockets, distillation, air separation, refrigeration and power cycles. In this paper, CFD analysis of heat transfer characteristics in nucleate pool boiling of saturated water in atmospheric conditions is performed in order to find out the influence of heat flux intensity on pool boiling dynamics. The investigation is carried out for four cases of different heat flux intensities and obtained results for velocity fields of liquid and void fractions are discussed. Grid independent test is also performed to improve the accuracy of calculation. In this way, complete picture of two-phase mixture behaviour on heated wall is represented.

Keywords: CFD analysis, two-phase mixture, void fraction

1. Introduction

Boiling is a phase-change phenomenon which draws attention for more than half a century. The main reason for this is that it presents the most efficent mode of heat transfer mechanism with very high heat transfer coefficients that could be achieved. It has numerous applications in industry, with one drawback, appearance of boiling crisis which constrains the implementation of this efficient heat transfer mechanism. Boiling crisis defines maximum values of heat flux for every industrial system and has direct influence on necessary dimensions, working life and investments into this equipment. It can cause damage of heated surface leading to its overheating or melting. Therefore, it is very important to investigate these phenomena, experimentally or numerically and find out the safety issues of these industrial equipment to work properly and with high durability.

Despite the tremendous efforts of researchers, the full picture of this phenomenon is far from complete, due to the inconsistency of the conditions and characteristics of the materials during experiments and non-linearity of the phenomena present in process. There is a large number of empirical models, derived from experiments, each of them applicable only to the certain conditions for whom it is performed. Experimental studies have focused on investigation of influence of certain parameters on heat transfer during boiling, for instance, surface tension ((Najim 2016; Xu 2016); gravitational force (Zhang 2014); surface orientation (Dadjoo 2017), external fields (Abdollahi 2017), and boiling pressure (Choon 2006). According to the experimental study of Pioro (2004), the most influental parameters on boiling heat transfer are surface imperfections. He concluded that surface modification presents one of the most significant methods for heat transfer inhancement. On the other hand, there are numerous efforts to investigate this phenomenon numerically; He (2001); Luttich (2004), etc. These models rely on assumptions which accuracy cannot be always determined. Despite these models, our numerical model includes convective heat transfer from the boiling surface and takes into account micro-scale parameters of the surface such as nucleation site density, liquid wetting contact angle and the bubble growth time.

In this paper, computational fluid dynamics analysis of heat transfer characteristics in nucleate pool boiling of saturated water is performed in order to predict the influence of heat flux intensity on pool boiling dynamics. The investigation is carried out for four cases of different heat flux intensities and obtained results for velocity fields of liquid and void fractions are discussed. The computations are based on volume of fluid method coupled with two-phase flow phase change model. Grid independent test is also performed to improve the accuracy of calculation. In this way, complete picture of two-phase mixture behaviour on heated wall is represented. Developed numerical method represents significant contribution to the development of nucleate boiling research methods, regarding previous investigation methods were mostly experimental, while developed empirical methods could be reliably applied only within narrow range of flow and thermal parameters of importance for process. This model enables clear understanding of bubble dynamics and heat transfer characteristics of water boiling in atmospheric conditions, direct numerical simulation of boiling process, as well as quantification of the impact of certain parameters of two-phase flow and heating wall conditions on the creation and running of nucleate pool boiling.

2. Modelling approach and governing equations

In-house self-developed code written in programming language FORTRAN was used for implementation of newly developed numerical model. Computational domain incorporates the pool and the heated wall. The pool is filled with two-phase mixture and vapor fills the rest of pool above the mixture swell level. The experimental conditions are the same as in (Stojanovic et al. 2016) so they wouldn't be discussed, Fig. 1. Heat transfer from the heated wall to the boiling liquid is modelled as follows: (1) conjugate heat transfer between wall and liquid and (2) heat flux at the bubble footprint. Here presented numerical model was previously used for boiling curve prediction (Stojanovic, et al., 2016). On the places of the heated wall where there is no bubbles, it is assumed convective heat transfer unlike the zones on which bubbles appear, where is assumed conjugate conduction. Boundary conditions between heated wall and two-phase mixture are implemented into the model through bubble nucleation site density and bubble residence time, to predict the influence of these parameters on pool boiling dynamics.

In order to capture the motion of the liquid and vapor interface, the volume of fluid model coupled with two-phase change model is implemented. Transport equations are written for both phases. The energy transport equation is written only for liquid phase with the assumption that vapor is saturated. Continuity equation for the volume fraction is applied in order to track the interface between the liquid and vapor:

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \frac{\partial\left(\alpha_k \rho_k u_{k,i}\right)}{\partial x_i} = \left(-1\right)^k \left(\Gamma_e - \Gamma_c\right) \tag{1}$$

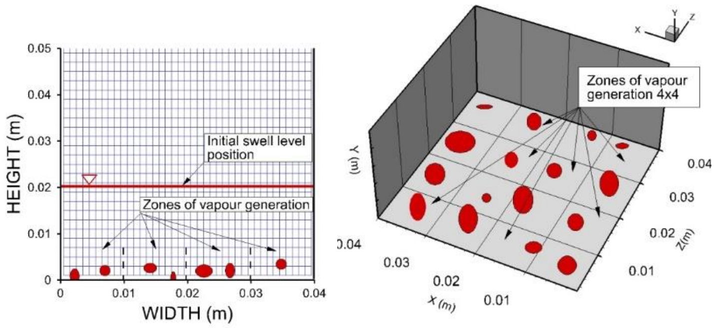

Fig. 1. Water pool (left) and zones of vapour generation (right), (Stojanovic et al., 2016).

Momentum equation is solved throughout the domain and can be written as:

$$\frac{\partial\left(\alpha_k \rho_k u_{k,i}\right)}{\partial t}+\frac{\partial\left(\alpha_k \rho_k u_{k,i} u_{k,j}\right)}{\partial x_i}=-\alpha_k \frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_i}\left[\alpha_k\left(\rho_k \nu_k \frac{\partial u_{k,i}}{\partial x_j}\right)\right]+ \tag{2}$$
$$\alpha_k \rho_k g_i+(-1)^k\left(\Gamma_e-\Gamma_c\right) u_{ik,i}+F_{21,i}$$

The energy equation can be written as:

$$\frac{\partial\left(\alpha_k \rho_k T_k\right)}{\partial t}+\frac{\partial\left(\alpha_k \rho_k u_{k,i} T_k\right)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\lambda_k}{c_{p,k}} \frac{\partial T_k}{\partial x_i}\right)+(-1)^k\left(\Gamma_e-\Gamma_c\right) T_k \tag{3}$$
$$+(2-\lambda) \dot{q}_b / c_{p,k}$$

where k=1 for liquid and k=2 for vapor. Source terms which define liquid-vapor interface processes are written on the r.h.s. of eq. (1)-(3). The rate of phase transition, denoted with Γ_e and Γ_c, represent the mass of water that evaporates per unit time and volume. Surface tension has an important role in the interfacial motion when describing bubble behaviour. It is denoted as $F_{21,i}$ and added as a source term to the momentum equation. Term q in energy equation represents a volumetric heat rate from the wall to the corresponding fluid phase per unit volume. For each control cell, the volumes of liquid and vapor sum to unity:

$$\alpha_1+\alpha_2=1 \tag{4}$$

The heated wall is modelled with following equation:

$$\frac{\partial T}{\partial t}=a \nabla^2 T+\frac{\dot{q}_h}{(\rho c)_w}-\frac{\dot{q}_b}{(\rho c)_w} \tag{5}$$

where q_h is a heat flux due to the external heat source in the wall (applicable, for instance, in case of wall electrical heating), while q_b denotes heat flux in control volumes where the bubbles grow and it represents heat sink. T and a denote the wall temperature and the thermal conductivity, respectively. For solving the set of transport equations (1) – (3), the control volume finite difference method is used. Flow field is discretized in Cartesian coordinates. There are sixteen zones on the heated wall where the random function determines the location of a bubble. It is assumed that the bubble nucleation does not occur, i.e., the heat flux $\dot{q}_b$ equals zero, that means in our case, if the value of void fraction in the control volume on the wall surface is higher than

1-10^-5^. We have calculated the volumetric heat power $\dot{q}_b$ as the ratio of heat necessary for the bubble growth and the bubble growth time until departure as $\dot{q}_b = \pi D_b^3 \rho_2 r / (6\tau)$.

3. Results and discussion

As mentioned above, surface tension or interfacial drag force is important in describing bubble dynamics. It is denoted as $F_{21,i}$ and added as a source term to the momentum equation. It is calculated as in Fritz (1935):

$$F_{21,i} = \frac{3}{4}\alpha_2\rho_1\frac{C_D}{D_P}\sqrt{\sum_{j=1}^{3}\left(u_{2,j}-u_{1,j}\right)^2}\left(u_{2,i}-u_{1,i}\right) \tag{6}$$

where C_D is the interfacial drag coefficient, and D_P is the diameter of the dispersed particle. The correlation for the interfacial drag coefficient C_D is given as follows:

$$C_D = 1,487 D_P\left(\frac{g\Delta\rho}{\sigma}\right)^{1/2}\left(1-\alpha_2\right)^3\left(1-0,75\alpha_2\right)^2 \tag{7}$$

Intensities of evaporation and condensation rate are dependent on relaxation time of evaporation τ_e and condensation τ_c ($\tau_e = \tau_c = 0.02s$) and are given as follows:

$$\Gamma_e = k_e\alpha_1\left(T_1 - T_2^{"}\right) = \frac{\alpha_1\rho_1}{\tau_e}\left(T_1 - T_2^{"}\right) \tag{8}$$

$$\Gamma_c = k_c\alpha_1\left(T_2^{"} - T_1\right) = \frac{\alpha_1\rho_1}{\tau_c}\left(T_2^{"} - T_1\right) \tag{9}$$

Results of simulation are shown in Figs. 2 - 5. Figure 2 shows a three-dimensional (3D) view at void fraction distribution in pool boiling for three different time intervals. The onset and further development of the process of nucleate boiling is clearly seen in the picture. Vapour bubbles are formed at discrete sites on the heated surface. The frequency of bubble release and number of active nucleation sites increase with the heat flux. For the first time interval, at t=0.9 s swell level is almost plane and its value is about 0.03 m. Only a small number of departed bubbles can be observed. For the second time interval, at t=1.9 s, the swell level position is higher and typical bursting of large vapour lumps is observed at the swell level. In addition, the formation of vapour layers at certain parts of the bottom heating wall is observed. At the third time interval, for t=2.1 s, the picture shows a large number of vapour bubbles through the whole volume of vessel, which are formed due to intensive vapour generation, as well as due to the merging of several vapour lumps into the larger one. In the right-hand part of Fig. 3, a horizontal cross section through the simulated pool vessel is shown. Therefore, the formation of vapour bubbles through the time can be observed.

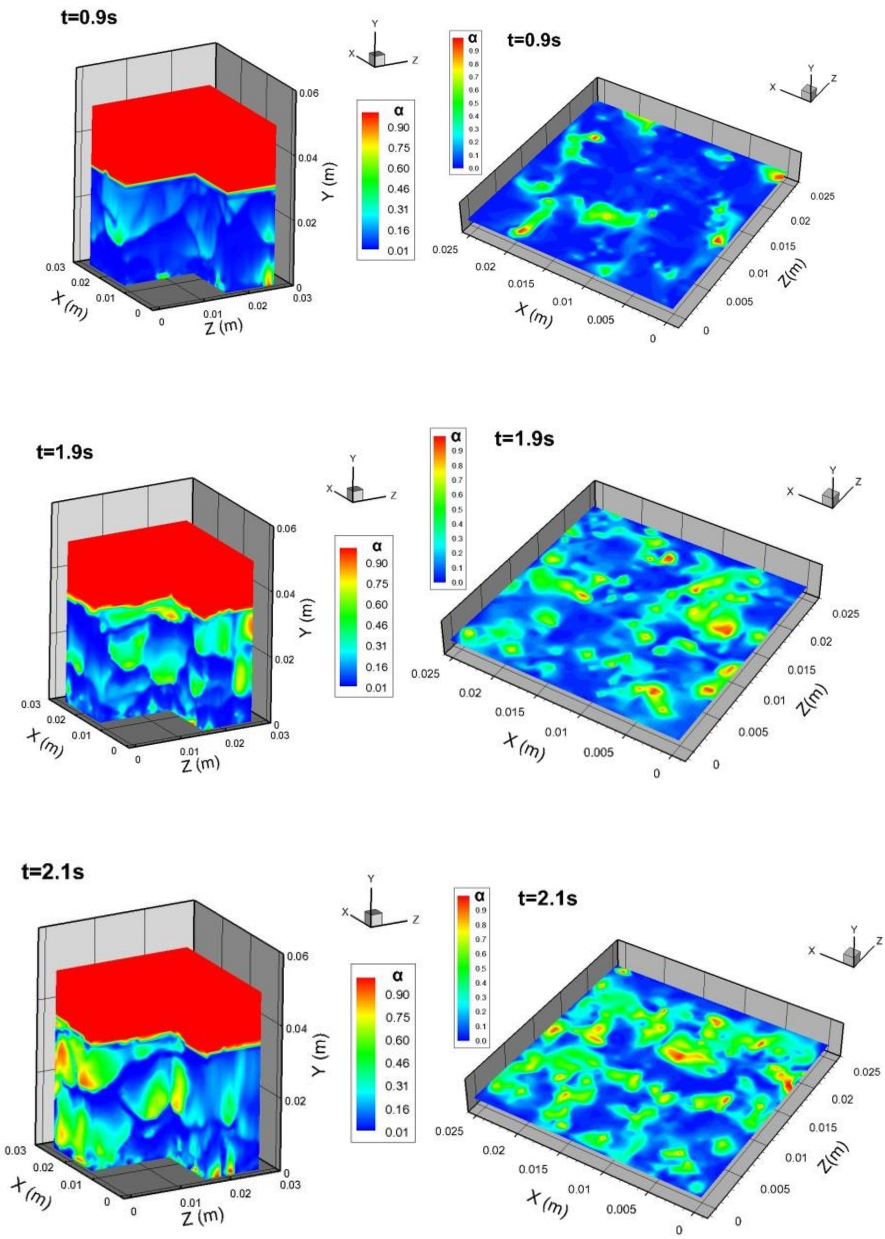

Fig. 2. 3D view of void fraction at the pool boiling for different time intervals (left) and horizontal cross section through the pool (right).

This is very important result regarding that after reaching maximum values of heat flux on the wall, the process of heat transfer will stop and only vapor will cover the whole surface of the wall. In that case, overheating of the wall surface will occur and dangerous conditions could appear. These unfavourable conditions can be predicted with our model and therefore, improve the performance of the industrial equipment.

Figures 3 and 4 show the velocity field for liquid and vapour fraction respectively, through the vertical cross section of the pool vessel, for four values of heat flux, namely 200 kW/m^2, 500 kW/m^2, 700 kW/m^2 and 1400 kW/m^2. The figures show variations of velocity through the pool vessel volume and at spots on the heated surface where bubbles are formed. These are the spots of heat sink at the top surface of the heated wall, because heat is conducted to the locations of the bubbles' growth.

Numerical tests with a denser grid (mesh sensitivity study) are performed to improve the accuracy of calculation using smaller dimensions of the cell's grid. Namely, when the grid is denser, time step becomes smaller and then time and spatial discretization errors are reduced. A numerical test with a double denser grid was performed, i.e., with eight times the number of control volumes that is 80x140x80 compared to a grid of 40x70x40 control volumes. The results showed that the difference between the mean wall temperatures of the wall overlap between the grid with dense and coarse mesh is less than 0.5K. Moreover, there were no differences in the behaviour of the two-phase mixture and temperature field on the wall for these two cases, as can be seen from Fig. 5, which shows change in the mean temperature of the heated wall for a 200 kW/m^2 thermal flux for these two grids.

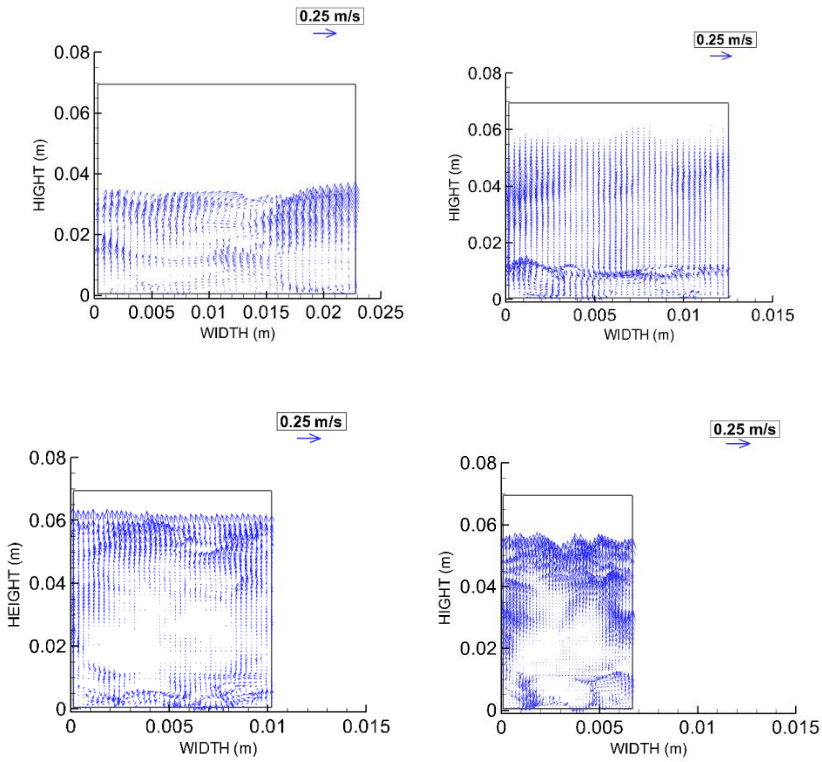

Fig. 3. Velocity field for liquid fraction; (a) q=200 kW/m^2; (b) q=500 kW/m^2; (c) q=700 kW/m^2; (d) q=1400 kW/m^2.

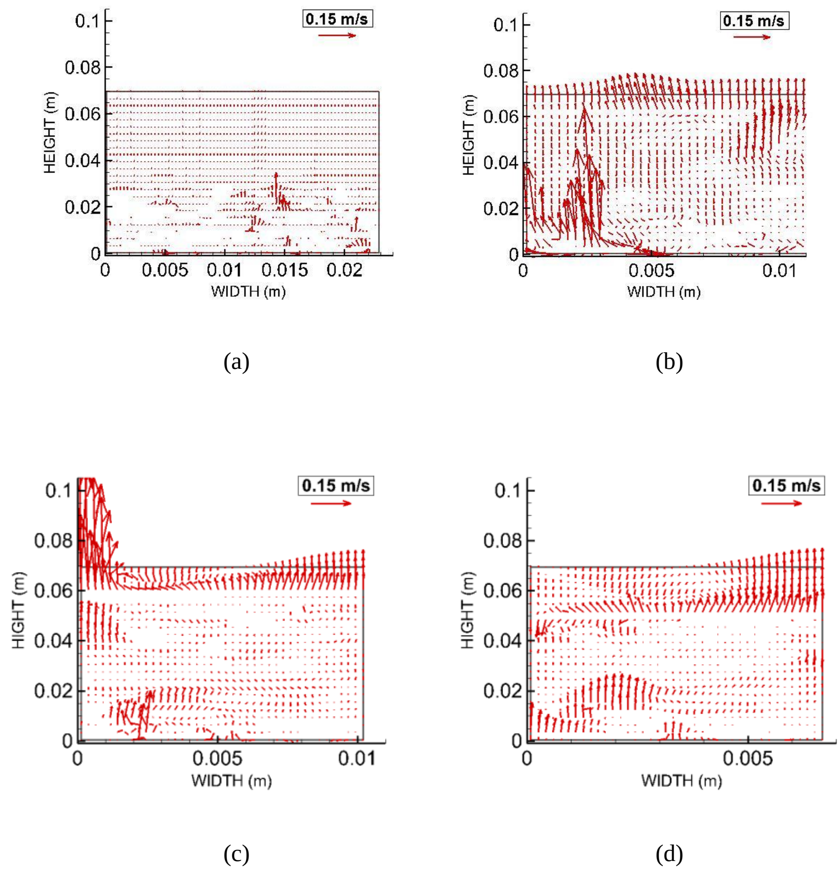

Fig. 4. Velocity field for void fraction: (a) q=200 kW/m^2; (b) q=500 kW/m^2; (c) q=700 kW/m^2; (d) q=1400 kW/m^2.

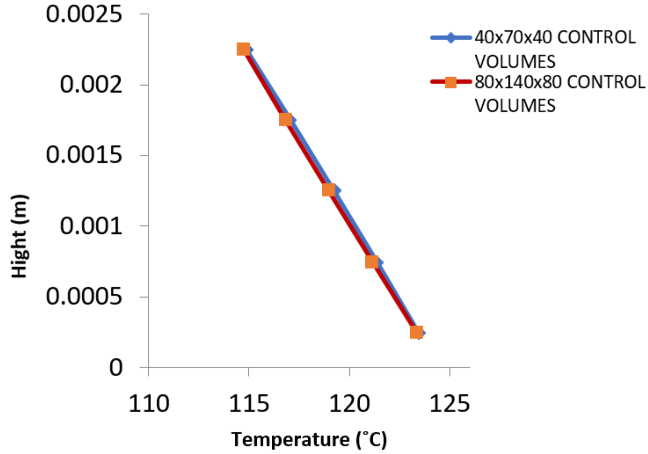

Fig. 5. Variation of mean temperature of heated wall with height of heated wall (average values across the horizontal planes) for heat flux value 200 kW/m^2.

4. Conclusions

Computational fluid dynamics methods are reliable tool for two-phase flow prediction unlike expensive experiments which are limited to the conditions they are performed for. This paper presents CFD analysis of nucleate pool boiling for different heat flux values. Self-developed mathematical and numerical model is implemented into in-house code written in FORTRAN. This model can accurately predict thermal-hydraulic conditions on the heated wall of two-phase vapor-liquid mixture. Results of the simulations for the process of nucleate boiling during time and velocity fields of vapor and liquid fractions are presented. The primary objective of this study is to assess the suitability of developed model for simulation of complex process of boiling. It shows that using appropriate input parameters, this model can be further implemented for more complex systems to reliably predict velocity fields under different heat fluxes. Obtained results contribute to valuable insights for practical hot water boilers and heat exchanger designers as well as those interested in numerical methods for nucleate pool boiling heat transfer prediction.

Acknowledgements The research was funded by the Ministry of Education, Science and Technological Development of the Republic of Serbia.

References

Abdollahi A, Salimpour M.R, Etesami N (2017). Experimental analysis of magnetic field effects on the boiling heat transfer of a ferrofluid, Applied Thermal Engineering, 111, 1101-1110.

Choon N K, Chakraboty A, Aye S M, Xiaolin W, (2006). New pool boiling data for water with copper foam metal at sub-atmospheric pressures: experiments and correlations, Applied Thermal Engineering, 26, 1286-1290.

Stojanovic A, Stevanovic V, Vladimir S, Petrovic M, Zivkovic D, (2016). Numerical investigation of nucleate pool boiling heat transfer, Thermal Science, 20, Suppl. 5, pp. S1301-S1312

Dadjoo M, Etesami N, Esfahany N.M, (2017). Influence of orientation and roughness of heater surface on critical heat flux and pool boiling heat transfer coefficient of nanofluid, Applied Thermal Engineering, 124, 353-361.

Fritz W (1935). Calculation of Maximal Bubble Volume (in German language), Physikalische Zeitschrift, 36, 379–384.

He Y, Shoji, M, Maruyama, S (2001). Numerical study of high heat flux pool boiling heat transfer, International Journal of Heat and Mass Transfer, 44, 2357-2373.

Ishii M (1987). Two-fluid Model for Two-phase Flow, 2nd Int Workshop on Two-Phase Flow Fundamentals, Rensselaer Polytechnic Institute, USA.

Luttich T, Marquardt W, Buchholz M, Auracher H (2004). Towards a unifying heat transfer correlation for the entire boiling curve, International Journal of Thermal Sciences, 43, 1125-1139.

Najim A, Pise S (2016). Boiling heat transfer enhancement with surfactant on the tip of a submerged hypodermic needle as nucleation site, Applied Thermal Engineering, 103, 989-995.

Pioro I, Rohsenow W, Doerffer, S (2004). Nucleate pool boiling heat transfer: review of parametric effects of boiling surface, International Journal of Heat and Mass Transfer, 47, 5033-5044.

Theofanous T G (2002). The Boiling Crises Phenomenon, Experimental Thermal and Fluid Science, 26, 775-792.

Xu Z, Zhao C (2016). Enhanced boiling heat transfered by gradient porous metals in saturated pure water and surfactant solutions, Applied Thermal Engineering, 100, 68-77.

Zhang Y, Wei J, Xue Y, Kong X, Zhao J (2014). Bubble dynamics in nucleate pool boiling on micro-pin-finned surfaces in microgravity, Applied Thermal Engineering, 70, 172-182.