E-ISSN: 2664-8784
P-ISSN: 2664-8776
Impact Factor: RJIF 8.26
IJRE 2025; 7(2): 194-197

www.engineeringpaper.net
Received: 01-09-2025
Accepted: 02-10-2025

Do-Ik Ri
Faculty of Thermal Engineering, Kim Chaek University of Technology, Pyongyang, Democratic People's Republic of Korea

Guk-Chol Ma
Faculty of Thermal Engineering, Kim Chaek University of Technology, Pyongyang, Democratic People's Republic of Korea

Chol Jin Jon
Faculty of Thermal Engineering, Kim Chaek University of Technology, Pyongyang, Democratic People's Republic of Korea

Corresponding Author:
Chol Jin Jon
Faculty of Thermal Engineering, Kim Chaek University of Technology, Pyongyang, Democratic People's Republic of Korea

CFD simulation of boiling heat transfer at flat electrodes of high voltage steam boiler

Do-Ik Ri, Guk-Chol Ma and Chol Jin Jon

DOI: https://www.doi.org/10.33545/26648776.2025.v7.i2b.145

Abstract
CFD simulations of boiling heat transfer characteristics were carried out according to the geometry of the flat plate electrode placed horizontally in a high-voltage steam boiler. Through CFD simulation, the reasonable geometry was determined.

Keywords: Steam boiler, high voltage, boiling heat transfer, CFD simulation

Introduction

1. Mathematical model

1.1 Governing equation

To simulate the water boiling process, an Eulerian multiphase model coupled with wall boiling model was used. The individual phases applied in the multiphase model are both vapor (water vapor) and liquid (water) phases, which are continuous phases.

Hence, the Euler multiphase model is in particular the Euler-Euler two-phase model.

Then, for each phase, we have to solve the continuity, momentum and energy equations, respectively [1].

1.2 The Interface Area Concentration (IAC) Model

To calculate the boiling heat transfer coefficient accurately, the characteristics of bubbles after separation from the electrode surface should be considered.

Therefore, we used the IAC (Interfacial Area Concentration) model to consider the mechanism of coalescence-collapse and evaporation-condensation of droplets.

The interface area concentration is defined as the interfacial area between two phases for a unit mixing volume. This is an important factor in predicting mass, momentum and energy transfer through the interface between phases [2]. As pointed out in Alali *et al.* [3], the transport equation for IAC derived by Yao and Morel [4] is

$$\frac{\partial \chi_b}{\partial t}+\nabla\cdot\left(\vec{v}_v\chi_b\right)=\frac{2}{3}\frac{\chi_b}{\alpha_v\rho_v}\left(m-\alpha_v\frac{\partial\rho_v}{\partial t}\right)+\varphi_1+\varphi_2+\varphi_3 \tag{1}$$

Where χ_b is the boundary area concentration, $\vec{v}_v$ is the velocity of vapor phase, ρ_v is the density of vapor phase α_v is the volume fraction of vapor, ϕ_1 is the source term considering the coalescence phenomenon due to random collisions, ϕ_2 is the source term considering the bubble breakdown phenomenon due to turbulence effects, and ϕ_3 is the source term considering bubble formation. When bubbles are removed from the electrode surface, the temperature of the vapor inside the bubble is kept lower than the temperature of the vapor, while the temperature of the supercooled liquid is kept lower than that of the vapor. Due to the presence of such a temperature difference, condensation of vapor at the interface takes place, which is denoted by m.

We used Yao and Morel models [4] to take into account the coupling and collapse source terms as in Alali et al, [3].

1.3 Wall boiling model

We used the RPI model of Kurul and Podowski [5] as wall boiling model. In the RPI model, the total heat flux (q_t) from the heating wall to the fluid is equal to the sum of the convective heat flux (q_c), quenching heat flux (q_q), and evaporative heat flux. From the work of Kurul and Podowski [3], the total heat flux is as follows:

$$q_t = q_c + q_q + q_e \tag{2}$$

Generally, the electrode surface can be divided into a portion A_b covered by bubbles and a portion covered by a liquid (1-A_b).
Then the convective heat flux of the liquid is

$$q_c = h_c\left(T_w - T_l\right)\left(1 - A_b\right) \tag{3}$$

Where h is the convective heat transfer coefficient of the liquid phase and T_w and T_l are the wall and liquid temperatures, respectively. The influence area A_b is defined based on the bubble separation diameter and bubble position density as follows.

$$A_b = \min\left(1, K\frac{N_b \pi D_s^2}{4}\right) \tag{4}$$

Where N_b is the bubble position density and D_s is the droplet escape diameter.
The empirical constant K is given by the Del Valle and Kenning relation [6]:

$$K = 4.8e^{\left(\frac{Ja}{80}\right)} \tag{5}$$

Here, J_a is defined as the Jacobian number as [4].

$$Ja = \frac{\rho_l c_{p,l}\left(T_w - T_l\right)}{\rho_v r} \tag{6}$$

The bubble position density in ANSYS Fluent can be calculated using the relations Lemmert and Chawla [7] or Kocamustafaogullari and Ishii [8]. Many previous studies have used Lemmert and Chawla relations in their work [4]. Unfortunately, we have found that the convergence of the solution is difficult using Lemmert and Chawla relations. Therefore, we used the Kocamustafaogullari and Ishii [8] relations to calculate the bubble position density.

$$N_b = \frac{1}{D_s^2}\left[\frac{2\sigma T_s}{\left(T_w - T_s\right)\rho_v r}\right]^{-4.4} f\left(\rho^*\right) \tag{7}$$

Where $f\left(\rho^*\right)$ is a function of density ratio: $\rho^* = \frac{\rho_l - \rho_v}{\rho_v}$
The droplet escape diameter is:

$$D_s^2 = 2.64\times10^{-5}\theta\left[\frac{\sigma}{\left(\rho_l - \rho_v\right)}\right]^{0.5}\rho^{*0.9} \tag{8}$$

Where a is the contact angle, whose value is 0.722 rad.
The fast cooling heat flux indicates that the hot wall is cooled by the entrainment of a low temperature ambient liquid after the bubble drops off the hot wall.
Seeing Alali *et al.* [3], the quench heat flux equation derived by Mikic and Rohsenow is:

$$q_q = 2C_d\sqrt{\frac{\lambda_l \rho_l c_{p,l} f_b}{\pi}}\left(T_w - T_l\right)A_b \tag{9}$$

Here, C_d is the bubble generation delay coefficient, which is proposed to represent the delay time between successive bubble separations.

In Eq. (9), f_b is the droplet escape frequency calculated by the Cole relation.

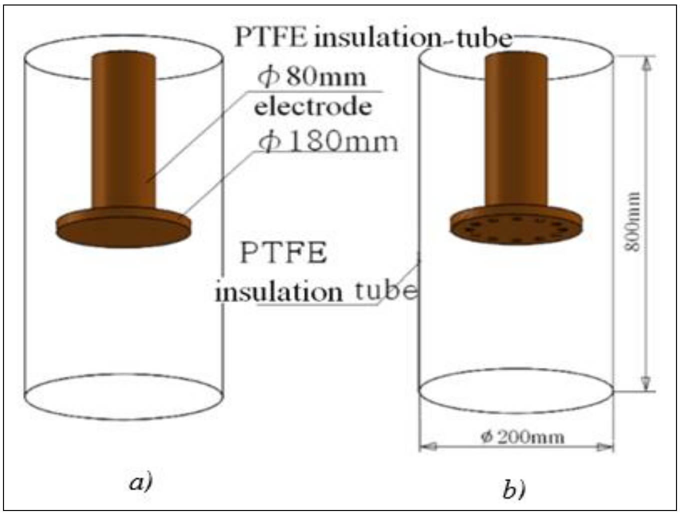
$$f_b = \sqrt{\frac{4g\left(\rho_l - \rho_v\right)}{3\rho_l D_s}} \tag{10}$$

As pointed out by Alali *et al.* [3], the evaporative heat flux induced by Bowring is:

$$q_e = \frac{\pi}{6}D_s^3 f_b N_b \rho_v r \tag{11}$$

2. Geometric model and mesh division

For CFD modeling of water boiling process at the electrode surface of a high-voltage steam boiler, the computational domain is first identified and based on it a geometric model is established.
In this paper, the corresponding geometric models are constructed as shown in Fig. 1 to compare the boiling heat transfer characteristics of the case of the absence and presence of holes in the disc-shaped electrode located at the bottom of the PTFE insulation tube.
The electrode diameter is 180 mm and the thickness is 15 mm. The material of the electrode is stainless steel, and the remaining area of the calculation area except the rod and electrode is the mixing area of the liquid and vapor phases (water and water vapor).
The water saturation pressure was 0.5 MPa, and all the material properties of the liquid and vapor phases were determined using NIST Standard Reference Database 23, Version 9.0.



a)- no hole case, b)- hole case

Fig 1: Electrode geometry

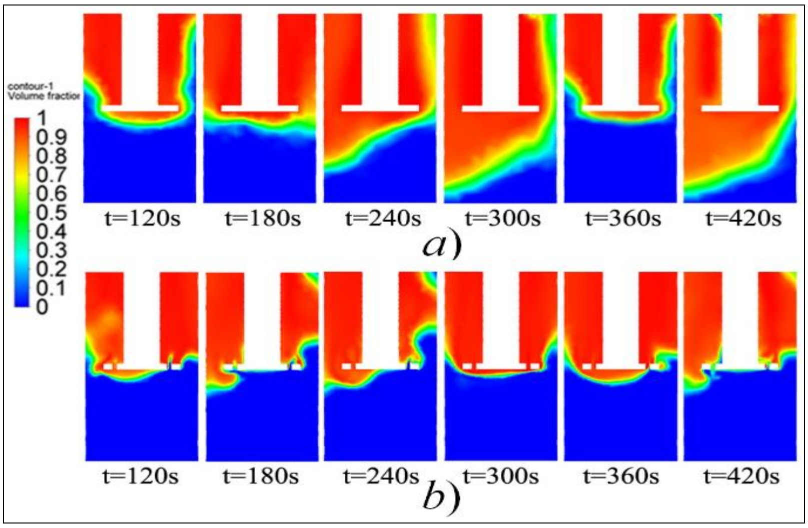
Both liquid and vapor phases were treated with incompressible fluids. The mesh for the constructed geometric models was performed using ANSYS Meshing 19.3.

3. Effect of electrode geometry on boiling heat transfer characteristics of water

3.1 Effect of electrode geometry on vapor volume fraction: Fig. 2 shows the boiling process of water (volume fraction change of water and steam) with time for the case of no hole in the disc (number of holes 18).

As can be seen in Fig. 2-a, if the electrode has no hole, the vapor at the bottom of the disc is collected in large amounts and then it is increased to a considerable extent to escape upward as the vapor pressure at the bottom of the electrode rises. This causes the electrode to be inadequate to contact with water, and the boiling of water is not sufficient, and the surface temperature of the electrode increases, leading to an increase in current. On the other hand, if the collected steam is instantaneously drained from the top, the electrode surface will suddenly come into contact with water and the temperature will drop, which causes a sudden surge of current.

Fig. 2-b shows that the presence of a hole in the electrode does not cause accumulation of steam at the bottom of the electrode and the boiling process is relatively stable. In this case, there is no significant change in the surface temperature of the disc compared to the case of no holes in the electrode, and thus the current fluctuations are relatively small.



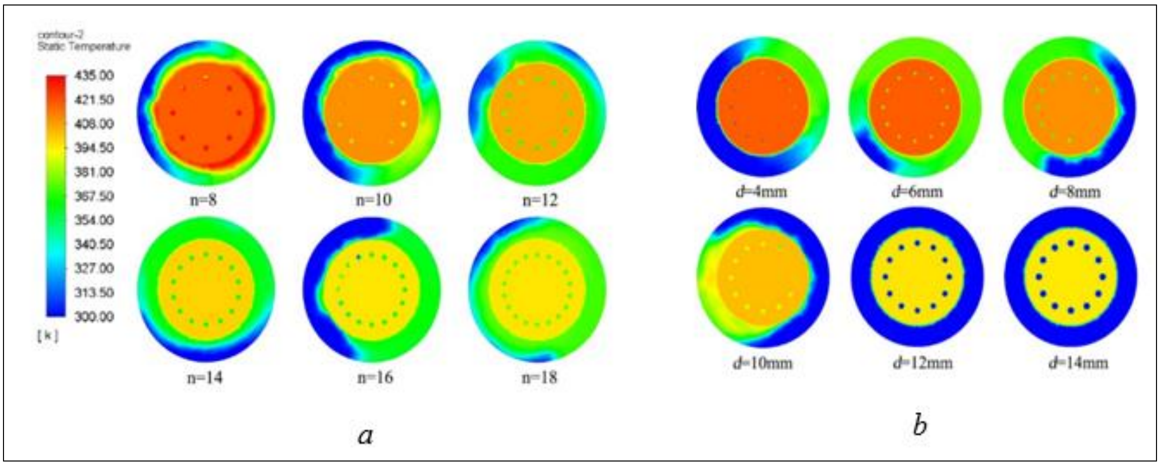
a)- no hole case, b)- hole case

Fig 2: CFD simulation results of water boiling process with time

3.2 Effect of electrode geometry on vapor volume fraction

3.2.1 Effect of number of holes

Fig. 3-a shows the variation of the average heat transfer coefficient at the electrode surface with the number of holes drilled on the electrode. The diameter of the hole was fixed to 8 mm. The value of the heat transfer coefficient increases rapidly at first, but the increase width decreases very much after the number of holes exceeds 16. It indicates that there is no significant change in the temperature of the electrode bottom from the number of holes up to 16, and consequently the rate of increase of the heat transfer coefficient decreases.



a) Temperature variation with number of holes, b) Temperature variation with hole diameter

Fig 3: Temperature variation at the bottom of the electrode at t = 420 s

3.2.2. Effect of holes diameter

Fig. 3-b shows the variation of the average heat transfer coefficient at the electrode surface with the diameter of the hole drilled in the electrode. The number of holes was fixed at 12.

The value of the heat transfer coefficient first increases rapidly, and after 12 mm of the hole diameter, the increase is not significant. This is because there is no significant difference in the change in the bottom surface temperature of the electrode from the hole diameter up to 12 mm.

4. Conclusion

In this paper, a solution for enhancing boiling heat transfer by rapidly draining the vapor formed at the bottom of the electrode is proposed and its analysis is carried out using ANSYS FLUENT 19.3, a heat and fluid flow analysis tool. For the analysis, first, the considered area containing electrode and water was identified, a mathematical model was developed to describe the boiling of water in a container, and, on this basis, CFD analysis of the boiling process of water was carried out. The results obtained are as follows.

First, increasing the number and diameter of holes is beneficial to reduce the amount of vapor deposited on the bottom of the electrode, prevent excessive overheating of the electrode, eliminate the fluctuation of current, and prevent the increase of the saturation temperature due to the increase of the pressure of the vapor.

Second, increasing the number and diameter of the holes is beneficial in increasing the heat flux density and heat transfer coefficient by enhancing the boiling heat transfer of water.

Third, with an excessive number of holes, the steam generation rate increases rapidly with an increase in area, resulting in a decrease in boiler stability, and with an excessive increase in the diameter of the holes, the area becomes too small to provide enough steam production.

From the above results, it is concluded that the optimum number and diameter of the holes must be determined to prevent current fluctuations, enhance heat transfer, ensure the boiler stability and ensure steam production.

Declaration of Competing Interests

The authors have no financial conflict for this paper.

Acknowledgments

This work was supported by Kim Chaek University of Technology, Democratic People's Republic of Korea. The supports are gratefully acknowledged. The authors express their gratitude to the editors and the reviewers for their helpful suggestions for improvement of this paper.

References

1. Höhne T, Krepper E, Montoya G, Lucas D. CFD simulation of boiling in a heated pipe including flow pattern transitions using the GENTOP concept. Nucl Eng Des. 2017;322:165-176.
2. Lucas D, Prasser H-M. Steam bubble condensation in sub-cooled water in case of co-current vertical pipe flow. Nucl Eng Des. 2007;237:497-508.
3. Alali A, Schöffel PJ, Herb J, Macian R. Numerical investigations on the coupling of the one-group interfacial area transport equation and subcooled boiling models for nuclear safety applications. Ann Nucl Energy. 2018;120:155-168.
4. Yao W, Morel C. Volumetric interfacial area prediction in upward bubbly two-phase flow. Int J Heat Mass Transf. 2004;47:307-328.
5. Kurul N, Podowski MZ. Multidimensional effects in forced convection subcooled boiling. In: Proceedings of the 9th International Heat Transfer Conference; 1990; Jerusalem. p. 21-26.
6. Del Valle MVH, Kenning DBR. Subcooled flow boiling at high heat flux. Int J Heat Mass Transf. 1985;28:1907-1920.
7. Lemmert M, Chawla JM. Influence of flow velocity on surface boiling heat transfer. In: Heat Transfer in Boiling. New York: Academic Press and Hemisphere; 1977. p. 237-247.
8. Kocamustafaogullari G, Ishii M. Foundation of the interfacial area transport equation and its closure relations. Int J Heat Mass Transf. 1995;38:481-493.