Transaction of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 28-30, 2015

Preliminary Study of ONB in Narrow-Vertical Rectangular Channel

Omar S.AL-Yahia*, Daeseong Jo
School of Mechanical Engineering, Kyungpook National University, 80 Daehak-ro, Buk-gu, Daegu 702-701, Republic of Korea
**Corresponding author: omar.alyahia@knu.ac.kr*

1. Introduction

Subcooled boiling is referring to the phenomenon where the vapor bubbles can detach from the heated surface (the boiling occurs) while the average bulk temperature is below the saturation value. The location where the vapor bubble can first exist at the heated surface is called "onset of nucleate boiling (ONB). The subcooled boiling is highly efficient to remove the heat owing to the high heat transfer coefficient. The heat transfer is affected by the motion of the bulk liquid as well as the latent heat transport of the liquid microlayer between the bubble and the heated wall [1]. However, with increasing in the wall temperature, the bubble growth will increase and may they aggregate at the heated surface forming a vapor film, which will prevent the heat transport from the wall and that leads to highly rise in wall temperature. This phenomenon called departure from nucleate boiling (DNB).

Many experimental and numerical CFD methods were carried out to investigate the subcooled boiling because of its importance in the industrial applications [1, 2, 3 and 4]. In the present study, vertical narrow rectangular channel heated from both side was simulated by using CFX-14 to investigate the subcooled wall boiling, and identical simulation is done by using TMAP to compare the ONB location between numerical simulation and empirical correlations that implemented in TMAP.

2. Model and methods

The simulation was conducted on a 300 mm long narrow rectangular channel. The channel thickness and width are 2.35 mm and 60 mm respectively, as shown in Fig.1. The heater width is 50 mm. The 1 m/s is the velocity of the downward flow. The inlet temperature of the coolant is 40 °C at atmospheric pressure. The applied power is 10-16 kW. A uniform heat flux ranging from 333.3 kW/m^2 to 533.3 kW/m^2 was applied to the heated plates.

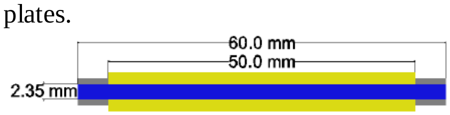

Fig1. Cross sectional view of the test section

2.1. TMAP model

TMAP is single phase, steady state 1-D thermal hydraulics code for plate type fuel [5]. The simplified one dimensional energy equation is used to determine the coolant temperature in the axial direction. The wall temperature is determined based on the convective heart transfer equation. Different heat transfer coefficient packages are implanted in TMAP. In this study, the simulation is done using two different correlations to predict the wall temperature; Dittus and Boelter

$$\mathrm{Nu} = 0.023Re^{0.8}Pr^{0.4} \tag{1}$$

The second package is new correlations developed at KAERI for narrow rectangular channel in both flow regimes; laminar and turbulent flow as shown in Eq (2) and Eq (3), respectively;

$$\mathrm{Nu}_L = 2.0129Gz^{0.3756} \tag{2}$$

$$\mathrm{Nu}_T = 0.0058Re^{0.9383}Pr^{0.4} \tag{3}$$

Bergles-Rohsenow correlation is used to determine an ONB heat flux and temperature as;

$$q"_{ONB} = 1082P^{1.156}[1.8(T_{ONB} - T_{sat})]^{\frac{2.16}{P^{0.0234}}} \tag{4}$$

2.2. CFX model

The set of equation solved by CFX-14 are the unsteady Navier-Stokes equations in their conservation form, which describe the processes of momentum, heat and mass transfer [6]. Multi-phase model is simulated to predict the ONB location through the flow channel. According to the RPI model, the total heat flux from the heated wall to the fluid is divided into three components; single phase convection heat flux q_c, the evaporation heat flux q_e and the wall quenching heat flux q_q.

$$q_w = q_c + q_e + q_q \tag{5}$$

Tolubinski and Kostanchuk formula is used to evaluate the bubble departure diameter [7]. The constant bulk bubbles mean diameter size of 1.2 mm [8]. The wall nucleation site density is given by *Lemmert Chawla* [9]. The bubble departure frequency is determined using *Cole* formula [10]. The volumetric heat transfer coefficient calculated by *Ranz-Marshall* model [11], where *Del Vall and Kenning* model is used to calculate the liquid quenching heat transfer coefficient [12].

The Shear Stress Turbulence (SST) model is used only for liquid phase. Sato enhanced eddy viscosity is use to take the bubble induced turbulence into consideration. The interfacial momentum transfer between liquid and vapor is modelled with the interfacial forces; drag and non-drag forces. The interfacial drag force is calculated according to the correlation of *Ishii and Zuber* [13]. The lift force coefficient is given by *Tomiyama* [14]. The wall lubrication force is given by *Antal* model [15]. *Favre Averaged Drag Force* is used to calculate the turbulent dispersion force.

3. Results

The numerical results using CFX-14 are discussed and compared with the results obtained from TMAP. The energy balance between TMAP and CFX are mostly identical as shown in Fig.2. The figure shows the average outlet bulk temperature for the used power range (10 kW – 16 kW). The coolant temperature distribution along the heated channel for 10 kW deposited power is shown in Fig.3. The coolant temperature increases gradually (linearly) in the downward direction owing to the uniform applied heat flux.

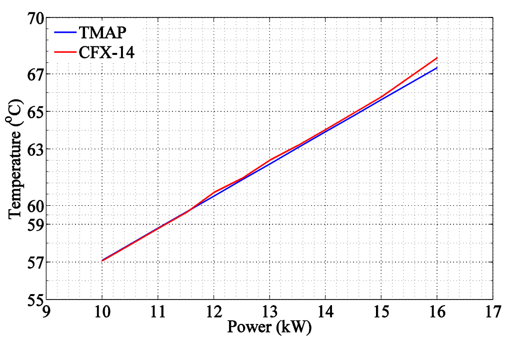

Fig.2 Outlet bulk Temperature

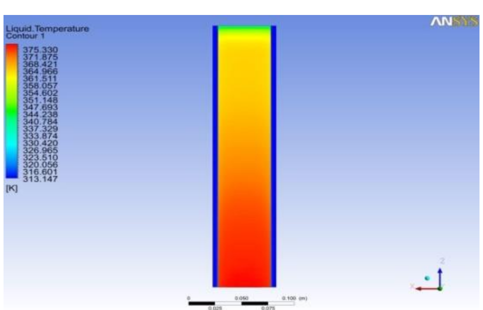

Fig.3 liquid temperature distribution

Fig.4 shows the average wall temperature evolution at different used power (10, 12 and 14 kW). In TMAP, the predictive wall temperature using KAERI correlation is always higher than that using Dittus-Boelter, which means that the single phase heat transfer coefficient using KAER correlations is lower than that using Dittus-Boelter. The results obtained by using KAERI correlation are always conservative. In CFX-14 analysis, the wall temperature varies with the power in between KAERI and Dittus-Boelter correlations because it takes into account the effect of two phase flow, not like TMAP that only for single phase modeling. With increasing in the power, the boiling occurs in the channel more early (near the inlet), which leads to enhance the heat transfer from the wall to the coolant. For that reason the wall temperature become closer to the one calculated using Dittus-Boelter correlation. And that shows the effect and benefits of the subcooled boiling phenomenon. The differences in the wall temperature explain the variation in the ONB location between the three simulations, as shown in Fig.5.

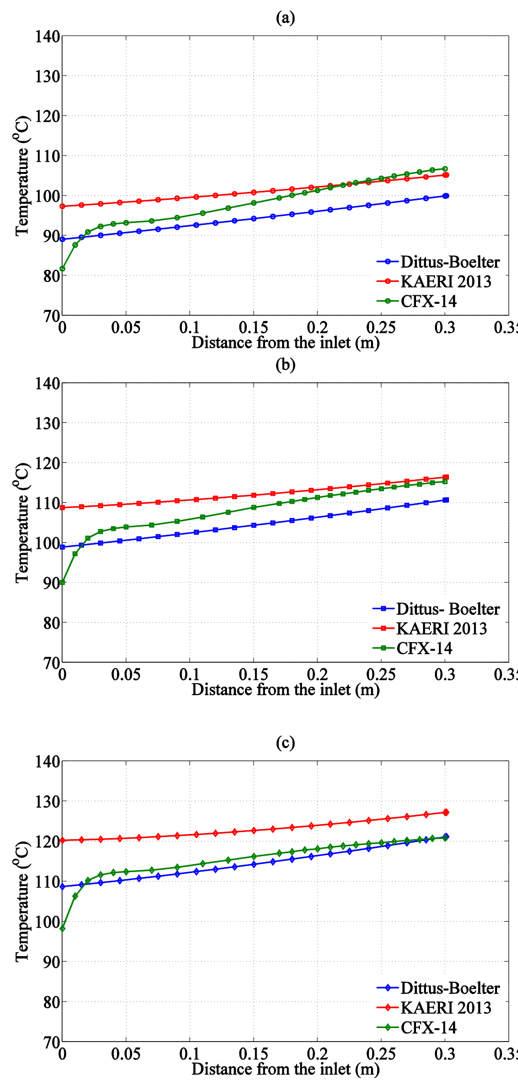

Fig.4. Wall Temperature; a) 10 kW b) 12 kW c) 14 kW

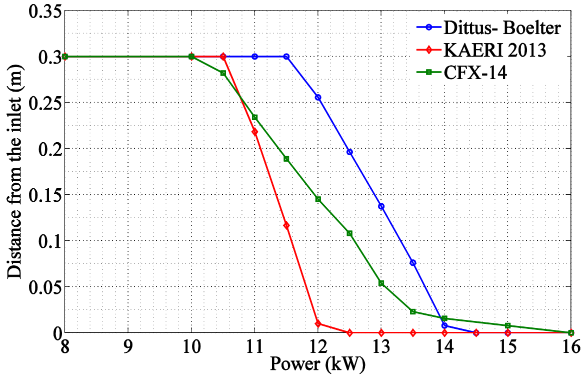

Fig.5 ONB location

REFERENCES

[1] Steiner, H., Kobor, A., Gebhard, L., A wall heat transfer model for subcooled boiling flow. Proceeding of the ASME-ZSIS International Thermal Science Seminar II. Bled, Slovenia, Jne 13-16, 2004.

[2] Zhang, R., Cong, T., Tian, W., Qiu, S., Su, G., Prediction of CHF in vertical heated tube based on CFD methodology. Progress in Nuclear Energy 78, 196-200, 2015.

[3] Podowski, M., Toward mechanistic modeling of boiling heat transfer. Nuclear Engineering and Technology, Vol.44 No. 8, 2012.

[4] Koncar, B., Matkovic, M., Simulation of turbulent boiling flow in a vertical rectangular channel with one heated wall.

[5] Jo, D., Park, J., Chae, H., Development of thermal hydraulic and margin analysis code for steady state forced and natural convective cooling of plate type fuel research reactors. Progress in Nuclear Energy, 71, 39-51, 2014.

[6] ANSYS CFX-Solver Theory Guide, Release 15.0, November 2013. pp.22.

[7] Tolubinsky, V.I., Kostanchuk, D.M. Vapour bubbles growth rate and heat transfer intensity at subcooled water boiling. In: Heat Transfer 1970, Preprints of papers presented at the 4th International Heat Transfer Conference, vol. 5, Paris (Paper No. B-2.8) 1970.

[8] Koncar, B., Krepper, E., CFD simulation of forced convective boiling in heated channel. Procceding of the workshop on Benchmarking of CFD Codes for Application to Nuclear Reactor Safety (CFD4NRS), Munich, Germany 5-7 Sep 2006.

[9] Lemmert, M., Chawla, J. Influence of flow velocity on surface boiling heat transfer coefficient. In: Hahne, E., Grigull, U. (Eds.), Heat Transfer in Boiling, 237. Academic Press and Hemisphere, New York and Washington DC, p. 247, 1977.

[10] Cole, R. A photographic study of pool boiling in the region of the critical heat flux. AIChE J. 6, 533–538, 1960.

[11] Ranz, W., Marshall, W. Evaporation from drops. Chem. Eng. Prog. 48, 141–146, 1952.

[12] Del Valle, V.H., Kenning, D. Subcooled flow boiling at high heat flux. Int. J. Heat Mass Transf. 28, 1907–1920, 1985.

[13] Ishii, M., Zuber, N. Drag coefficient and relative velocity in bubbly, droplet or particulate flows. AIChE J. 25, 843–855, 1979.

[14] Tomiyama, A. Struggle with computational bubble dynamics. In: Third International Conference on Multiphase Flow, ICMF'98, Lyon, France, June 8–12, 1998.

[15] Antal, S., Lahey Jr., R., Flaherty, J. Analysis of phase distribution in fully developed laminar bubbly two-phase flow. Int. J. Multiph. Flow 17, 635–652, 1991.