Fifth International Conference on Computational Methods for Thermal Problems
THERMACOMP2018, July 9-11, 2018, Indian Institute of Science, Bangalore, INDIA
N. Massarotti, P. Nithiarasu, Pradip Dutta and C. Ranganyakalu (Eds.)

Numerical Analysis of Critical Heat Flux during Subcooled Boiling for a Vertically Downward Flow

Sumanth Theeda[1], Siva Subrahmanyam Mendu[2], Rajeshwar Sripada[3], Veeredhi Vasudeva Rao[4]

[1,2](Department of Mechanical Engineering, MVGR College of Engineering, India 535005,

theedasumanth9@gmail.com, m.sivasubrahmanyam@mvgrce.edu.in)

[3,4](University of South Africa, Johannesburg, South Africa,

rajeshwar33482@gmail.com, v_vasudevarao@yahoo.com)

ABSTRACT

Two-phase flows are encountered in various applications in power plants, chemical plants and nuclear industry. The current day industries requirement is to operate these two-phase flow equipments over a wide range of operating conditions and is subjected to harsh environment driven by high temperature corrosive gases. This prompts for usage of expensive materials and thermal coatings to protect from tube burnout and thereby extend the life, without compromising on cost. In the context of modernization or up-gradation of the existing equipment, the transportation limits put several constraints on making the heat exchanger fatter or taller, thereby adding to the complexity. To make these heat exchangers compact without compromising on its efficiency, one should look for alternative ways. One of the ways of doing it is to increase the number of passes by routing tubes in such a way that the two-phase mixture travels downwards and upwards in a periodical manner. This brings in additional risks of premature tube burnout by reaching Critical Heat Flux limit, two-phase flow instabilities and other potential risks, especially when the flow takes place in a vertically downward direction. Critical Heat Flux (CHF) or post burnout refers to the sudden decrease in heat transfer coefficient for a surface on which evaporation or boiling occurs. Exceeding this heat flux causes the replacement of liquid adjacent to the heat transfer surface with a vapor blanket. This blanket acts as a barrier to heat flow from the heat dissipating body, resulting in possible catastrophic failure. Hence accurate estimation of CHF risk is a mandate not only from performance or life perspective, but also more important from safety perspective. Further, conducting numerous experiments to understand the flow patterns and the CHF in vertically downward two-phase flow is time consuming and expensive. All these constraints give an opportunity to leverage the numerical tools that are relatively less expensive and are quicker to estimate CHF.

The present study focuses on the numerical analysis carried in a vertically downward subcooled flow using finite volume based commercial software Fluent by ANSYS Inc. Rensselaer Polytechnic Institute (RPI) boiling model in Fluent is used to predict the void fraction and CHF in a vertically downward flow. Turbulence effects are modelled by Shear Stress Transport (SST) k-ω and k-ε models. Most of the numerical work carried till date on two-phase flows focused on flow in vertically upward direction or in horizontal direction. Hence, the numerical models are validated against the data provided in open literature for vertically upward flows. The same models are extended for vertically downward subcooled flows in present investigation and the results for upward and downward flows are compared.

Keywords: *Computational Fluid Dynamics, RPI Boiling Model, Vertically Downward Two-Phase Flow, Void Fraction, Critical Heat Flux.*

1. INTRODUCTION

Significant amount of experimental work was done on two-phase flows in last 50 years focusing on understanding the flow patterns, estimating the void fractions and determining the CHF limits for the safety. Most of the investigations in this field were primarily focused on understanding the fluid flow patterns and heat transfer in horizontal tubes, inclined tubes, and in vertical tubes with flow directed upwards and covering wide range of operating conditions. Hall & Mudawar compiled a database, which was based on previous investigators as well based on their own investigations [1, 2]. Bartolomei and Chanturiya [4] conducted the experiments with pressurized water. Their experimental data was used for validation by many researchers. DEBORA experiments [7] also mark their importance in evaluating the CHF for R-12 refrigerant at high pressures to 2.62 MPa. Recent developments in computational facilities encouraged researchers to solve the governing equations for two-phase flows numerically. Ribeiro *et al.* [3] had adopted the experiments of Bartolomei and Chanturiya [4] and solved them numerically using commercially available FLUENT CFD code. The results were in good agreement with experimental data. Naveen and Veluswamy [6] had reported numerical solutions for Sodium boiling using FLUENT CFD code. They validated their model with Bartolomei and Chanturiya [4], and with DEBORA experiments [7]. Vyskocil and Macek [5] documented the numerical results generated using NEPTUNE_V2 CFD code and found that their results were satisfactory. They reported that NEPTUNE code was not suitable for solving low pressure cases and high heat flux cases due to numerical instabilities.

However, there were only limited investigations carried on vertically downward flows, where the buoyancy of the bubbles competes with the gravity of the liquid, resulting in the most complex flow behavior. The investigations in vertically downward flow were focused primarily on understanding the flow pattern maps and in estimating the void fractions [8]. A few investigations were focused on understanding the CHF for vertically downward flows experimentally [9, 10, 11]. To the best of the author's knowledge, hardly any numerical investigations were carried in vertically downward subcooled flows numerically. This gives an opportunity to investigate the void fraction and CHF in a vertically subcooled flow using numerical tools. This was the motivating factor for current investigations.

2. NUMERICAL ANALYSIS

This section briefly discusses the numerical analysis method carried for current investigations.

Void Fraction Validation:

The models were validated with the experimental work done by Bartolomei *et al.* [4] for the void fraction. The geometry was modelled using the Design Modeller software available in ANSYS and the analysis was carried with FLUENT. A steady-state, axi-symmetric model was considered for current simulations. The properties were assumed to be temperature dependent. Meshing was done in ICEM CFD of ANSYS Inc. Grid independent study was carried before finalizing the simulations. Turbulence effects were modelled using k-ω turbulence model. Table 1 shows the geometric and operating conditions used for current simulations.

Table 1. Geometric and operating conditions for RPI model

Parameter	**Bartolomei *et al.* [4]**
Fluid	Pressurized Water
Length of Pipe	2 m
Inner Diameter	15.4 mm
Heat Flux	570 kW/m^2
Mass Flux	900 $kg/m^2/s$
Inlet Sub-cooling	60 K
Operating Pressure	4.5 MPa

Critical Heat Flux validation for upward flow and extension to downward flow

Critical Heat Flux for upward flow was validated with the work done by Hoyer [12]. The RPI boiling model available in Fluent was used in current investigations for validation. Table 2 shows the geometric and operating conditions used in the present simulation study.

Table 2. Geometric and operating conditions for CHF model

Parameter	Hoyer [12]
Fluid	Pressurized Water
Length of Pipe	7 m
Inner Diameter	10 mm
Heat Flux	797 kW/m^2
Mass Flux	1495 $kg/m^2/s$
Inlet Sub-cooling	10 K
Operating Pressure	7.01 MPa

3. RESULTS

Void Fraction & Wall Temperature Validation for Upward Flow

The vapour void fraction prediction with numerical tool is validated with the work done by Bartolomei and Chanturiya [4][1] for vertically upward flow. Circumferential average of vapor void fraction along the length of the tube is presented. Figure 1 shows the comparison of vapor void fraction between current numerical analysis and experimental data. As it is evident, the results obtained are in good agreement with the experimental values documented by Bartolomei and Chanturiya [4].

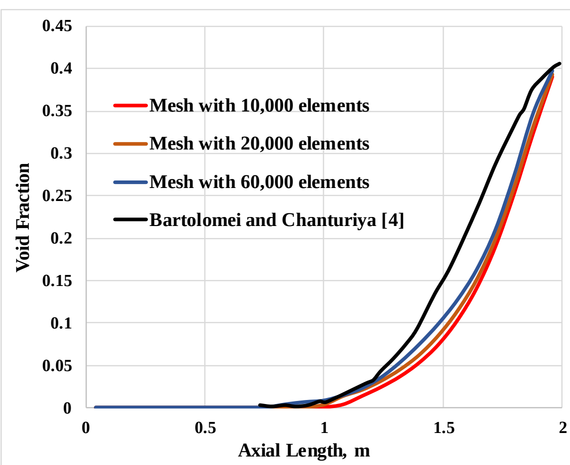

Fig. 1 Validation of vapor void fraction

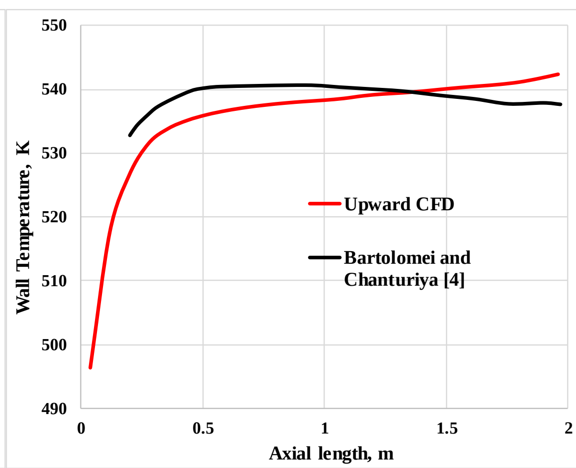

Fig.2 Validation of wall temperature

Figure 2 shows the comparison of wall temperature from current numerical analysis with the experimental data. The results show that the wall temperature prediction trends are in agreement

---

[1] Data taken from Riberio *et al* [3]

with the experimental data for upward flow. However, the wall temperature variations exist and is mainly due to the wall treatment procedures adopted in the turbulence modelling.

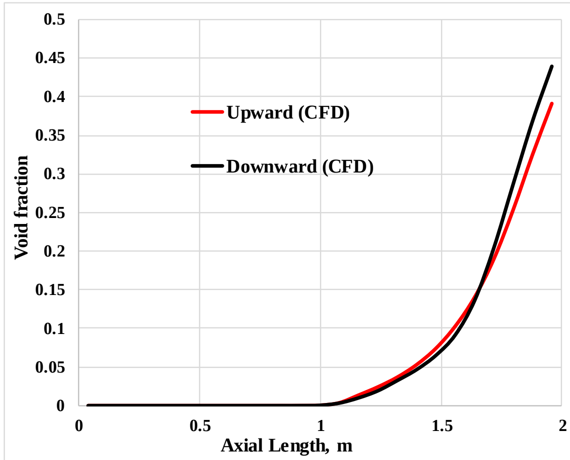

Fig. 3 Prediction of void fraction in downward flow

The validated void fraction model used for upward flow is extended to predict the void fraction for downward flow. The comparison of void fraction for upward and downward flows are shown in Fig. 3. The predicted results show that the void fraction at outlet is higher in downward flow than in upward flow, as expected and as reported most of the previous investigators.

Critical Heat Flux

This section presents the results obtained based on CHF investigations carried using Fluent. Fig. 4(a) shows the Critical Heat Flux validation for upward flow with experimental data in open literature [12]. As it is evident, the numerical model prediction of critical heat flux defined by sudden rise in wall temperature, is in good agreement with experimental data. Figure 4(b) shows the CHF prediction for upward flow and downward flow based on the extension of validated numerical upward flow CHF model. As it is observed from Fig. 4(b), there is hardly any difference in CHF for upward flow and downward flow at high mass fluxes and at high pressures.

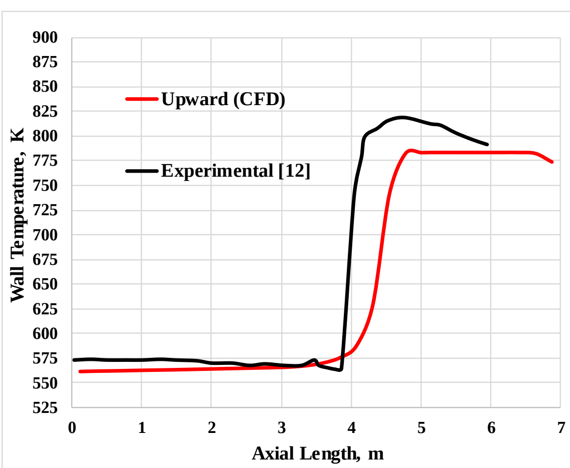

Fig. 4(a) Validation of CHF model with experiment

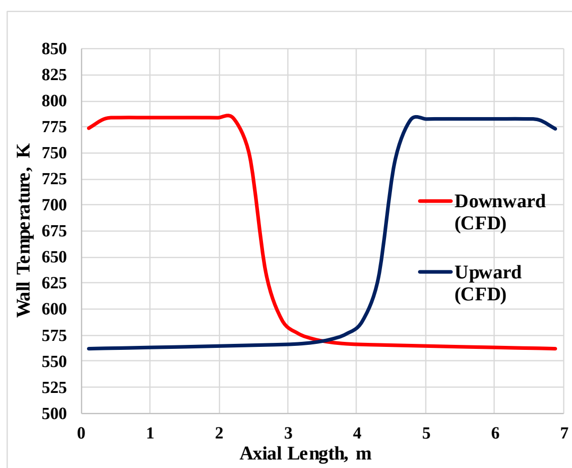

Fig. 4(b) Comparision of CHF in upward and downward flow

4. CONCLUSIONS

The numerical model for two-phase flow has been validated against the experimental data in open literature for void fraction, wall temperature and CHF in upward flow at high pressure. The numerical predictions are in good agreement with the experimental data for upward flow. These models are then extended to predict void fraction and CHF in a vertically downward subcooled flow. As expected, the void fraction at the outlet is higher for downward flow compared to upward flow, where as there is no significant change in CHF for upward and downward flow at high pressure and at high mass fluxes. This indicates that the flow direction has minimal impact at high mass fluxes and at high pressures. The available numerical models for vertically upward flow could be easily extended to model vertically downward flows at high mass fluxes. Further investigations are necessary to validate the suitability of these models for vertically downward two-phase flows with low mass fluxes and at low pressures.

5. ACKNOWLEDGEMENT

The authors acknowledge the support provided by Jagat Paridala, Ravi Kumar Sambangi, Akash Varma Rudraraju, Venkateswar Rao Ragina from MVGR College of engineering.

REFERENCES

[1] Hall, David D., and Issam Mudawar. "Critical heat flux (CHF) for water flow in tubes—I. Compilation and assessment of world CHF data." International Journal of Heat and Mass Transfer 43.14 (2000): 2573-2604.

[2] Hall, David D., and Issam Mudawar. "Critical heat flux (CHF) for water flow in tubes—II.: Subcooled CHF correlations." International Journal of Heat and Mass Transfer 43.14 (2000): 2605-2640.

[3] Ribeiro, Guilherme & Braz Filho, Francisco. “Assessment of Interfacial Heat Transfer Models Under Subcooled Flow Boiling.” (2017).

[4] Bartolomei, G. G., and V. M. Chanturiya. "Experimental study of true void fraction when boiling subcooled water in vertical tubes." Thermal Engineering 14.2 (1967): 123-128.

[5] Vyskocil, L., and J. Macek. "Boiling flow simulation in Neptune CFD and Fluent codes." XCFD4NRS, Grenoble, France (2008): 10-12.

[6] Raj, M. Naveen, and K. Velusamy. "CFD Simulation of Sodium Boiling in Heated Pipe using RPI Model.", Proceedings of the 2nd World Congress on Momentum, Heat and Mass Transfer (MHMT’17), Barcelona, Spain – April 6 – 8, ICMFHT, (2017).

[7] Garnier, J., E. Manon, and G. Cubizolles. "Local measurements on flow boiling of refrigerant 12 in a vertical tube." Multiphase Science and Technology 13.1&2 (2001).

[8] Bhagwat, Swanand Madhav. “Study of flow patterns and void fraction in vertical downward two phase flow”. Oklahoma State University, 2011.

[9] Ruan, S. W., G. Bartsch, and S. M. Yang. "Characteristics of the critical heat flux for downward flow in a vertical tube at low flow rate and low pressure conditions." Experimental thermal and fluid science 7.4 (1993): 296-306.

[10] Kaichiro, Mishima, Nishihara Hiedeaki, and Michiyoshi Itaru. "Boiling burnout and flow instabilities for water flowing in a round tube under atmospheric pressure." International journal of heat and mass transfer 28.6 (1985): 1115-1129.

[11] Shen, Zhi, et al. "Experimental investigation on heat transfer characteristics of smooth tube with downward flow." International Journal of Heat and Mass Transfer 68 (2014): 669-676.

[12] Hoyer, Norbert. "Calculation of dryout and post-dryout heat transfer for tube geometry." International journal of multiphase flow 24.2 (1998): 319-334.

[13] Sudo, Yukio, et al. "Experimental study of differences in DNB heat flux between upflow and downflow in vertical rectangular channel." Journal of Nuclear Science and Technology 22.8 (1985): 604-618.

[14] Mishima, Kaichiro, and Hideaki Nishihara. "Effect of channel geometry on critical heat flux for low pressure water." International journal of heat and mass transfer 30.6 (1987): 1169-1182.