14th Australasian Fluid Mechanics Conference
Adelaide University, Adelaide, Australia
10-14 December 2001

CFD Code Benchmark on Void Fraction Distribution in Subcooled Flow Boiling of a Concentric Annular Tube at Low Pressure

Denpong Soodphakdee[1], Jiyuan Tu[2], and Masud Behnia[3]

[1]Department of Mechanical Engineering
Khon Kaen University, Khon Kaen 40002, THAILAND

[2]Australian Nuclear Science and Technology Organisation (ANSTO)
PMB1 Menai, NSW 2234, AUSTRALIA

[3]School of Mechanical and Manufacturing Engineering
The University of New South Wales, Sydney NSW 2052, AUSTRALIA

Abstract

In a comparison exercise, three CFD codes, i.e. CFX-4.2, FLUENT 4.5, and RELAP 5/MOD2, were used for the simulation of subcooled flow boiling in a vertical concentric annular tube. The numerical predictions from these three codes were compared with experimental void fraction distribution. The available bubble size correlations were adopted into the two-fluid model of CFX-4.2. The adjustable parameters of the evaporation-condensation model of FLUENT 4.5 were modified. Surface heat flux, subcooled temperature, water flow rate and system pressures were varied to check the capability of each code at different conditions. The predicted void fraction distribution results shows that the 1D code RELAP 5 gives less accurate results compared with the modified general purpose codes. The more complex two-fluid model of CFX offers the most consistency of the void fraction distributions for the whole range of study, but under predict in the high subcooled region. The simple model of FLUENT predicts the void fraction in a relatively low Reynolds number range.

Introduction

In general one of the weak features of most CFD (Computational Fluid Dynamics) code is the simulation of multiphase flow. Boiling is a good example of a combination of two-phase flow with heat and mass transfer. The complexity of the boiling phenomena makes CFD simulation even harder. However, many general purpose CFD codes contain boiling models, but are yet to be widely used due to lack of validation. Validation of boiling simulation is hardly seen in the literature. Although experimentation has been the most common method of boiling research, cost and experimental uncertainty are very high. These factors have encouraged the development of CFD simulation in recent years.

Application of boiling heat transfer is now growing into the challenging field of electronic cooling and nuclear reactor analysis. The rapid growth of heat dissipation rate from the electronic devices drives the development of new techniques capable of keeping the operation temperature at a satisfactory level. The fuel element of nuclear reactors is a critical part for safety analysis. From the thermodynamics point of view, boiling is a constant temperature phenomena and therefore efforts are being taken to apply boiling heat transfer to the cooling of electronic devices. Because the mechanisms of boiling are not well understood, it is not yet applicable for boiling to be used as a normal electronic cooling practise. In addition the process of the design, prototyping and testing of electronic cooling devices is very short. For these reasons, CFD plays an important role in this field. Applying CFD to simulate boiling flow and heat transfer presents a challenge for mathematicians and engineers.

As mentioned earlier, implementing CFD for the simulation of multiphase flow is yet to be widely accepted, and validation of this complex heat transfer mode is necessary.

Evaporation heat transfer for cooling of electronic equipment was studied by Kristiansen *et al.* [2]. They highlighted that evaporative cooling is well suited for cooling high power electronic devices. Many reports in the literature are mainly experiment based. Analytical studies were also carried out in some boiling cases. Zhao *et al.* [7] studied the boiling mechanism in a narrow space analytically. The narrow spacing is suitable for electronic cooling because of the spacing limitation in electronic devices. The decreasing of channel hydraulic diameter results in an increasing heat transfer coefficient. Zhao *et al.* [7] also studied the growth of bubbles in a small gap between two horizontal parallel plates heated from the bottom plate. Stephens and Harris [3] presented a benchmark solution of numerical modelling using CFD code CFX-4.2 and a numerical code developed in MATLAB. The one-dimensional two-phase flow modelling of a horizontal uniform channel was solved using MATLAB. The results were then compared with the CFD simulation. The solution of that simple geometry and flow showed a perfect agreement with each other. However there is no comparison with experimental data demonstrated in that study. The comparison of CFD simulation with experimental data was later conducted by Tu [4]. The bubble mean diameter correlation in CFX-4.2 was replaced by the correlation developed by Zeitoun and Shoukri [5]. The effect of bubble size to void fraction distribution was studied.

Void Fraction in Subcooled Boiling

When subcooled liquid enters the heated portion in any system, the temperature distribution adjacent to the heated surface may result in the initiation of the subcooled boiling process. Subcooled boiling continues downstream but the void fraction cannot significantly increase because of high subcooling (see highly and moderately subcooled section of figure 1). The void fraction increase rapidly when the liquid has a slightly subcooled temperature.

Multiphase Flow Modelling for Subcooled Flow Boiling

The basic algorithm for modelling of subcooled boiling is the Eulerian multiphase model. Two sets of conservation equations of continuity and momentum for liquid and vapour phases are solved by the inclusion of volume fraction. The interaction between the phases is the transport of mass, momentum and energy need to be taken into account. In order to clarify the differences between CFX 4-2 and FLUENT 4.5, the interface modelling is briefly presented here.

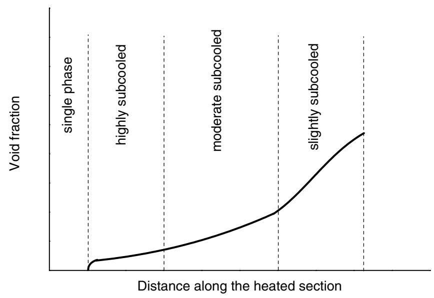

Figure 1: Void fraction in subcooled boiling

Drag force

Both FLUENT and CFX codes use the same algorithm to calculate drag force, which is

$$F_{lg} = \frac{3}{4} C_D \frac{\alpha_g \rho_l \left| \vec{u}_g - \vec{u}_l \right| \left(\vec{u}_g - \vec{u}_l \right)}{d_g} \tag{1}$$

where F, C_D, α, ρ, $\vec{u}$, and d are the drag force, drag coefficient, volume fraction, density, velocity, and bubble diameter, respectively. Subscripts l and g denoted the liquid and gas phases, respectively.

Heat transfer

Both FLUENT and CFX codes use the same method of calculation for heat transfer using fundamental formula.

$$Q_{lg} = H_{lg} \left(T_g - T_l \right) \tag{2}$$

where H_{lg} is the heat transfer coefficient between phases. H_{lg} is related to the bubble Nusselt number as

$$H_{lg} = \frac{6 k_l \alpha_g Nu_g}{d_g^2} \tag{3}$$

where k is the thermal conductivity.

Mass transfer

There are two types of mass transfer between phases;

unidirectional mass transfer The unidirectional model defines a positive mass flow rate per unit volume between phases.

evaporation–condensation FLUENT contains a simple phenomenological model for a mixture of two phases (liquid and vapour). The evaporation rate $\dot{m}_g$ and the condensation rate $\dot{m}_l$ are determined from

$$\begin{aligned} \dot{m}_g &= r_g \alpha_l \rho_l \tfrac{(T_l - T_{sat})}{T_{sat}} & T_l \geq T_{sat} \\ &= 0 & T_l < T_{sat} \end{aligned} \tag{4}$$

$$\begin{aligned} \dot{m}_l &= r_l \alpha_g \rho_g \tfrac{(T_{sat} - T_g)}{T_{sat}} & T_g \leq T_{sat} \\ &= 0 & T_g > T_{sat} \end{aligned} \tag{5}$$

The factors r_g and r_l are fixed to unity in CFX, while they can be adjusted in FLUENT.

Bubble diameter

CFX incorporates the ability to modify bubble diameter, d_g, to suit the specific case, if known. Bubble diameter correlations from previous studies may be used. The default algorithm implemented in the bubble diameter calculation in CFX is a linear relationship between bubble diameter and temperature [1].

$$d_g = \frac{d_1 \left(T_{sub} - T_0 \right) + d_0 \left(T_1 - T_{sub} \right)}{T_1 - T_0} \tag{6}$$

where T_{sub} is the local subcooling; d_0 and d_1 are the bubble diameters at reference liquid subcoolings T_0 and T_1. Outside this temperature range the diameters become constant. In order to facilitate a better prediction, for the cases shown in this paper, the bubble diameter was calculated using the empirical correlation [5].

$$\frac{d_g}{\sqrt{\sigma / g\Delta\rho}} = \frac{0.0683 \left(\rho_l / \rho_g \right)^{1.326}}{Re^{0.324} \left[Ja + \frac{149.2 \left(\rho_l / \rho_g \right)^{1.326}}{Bo^{0.487} Re^{1.6}} \right]} \tag{7}$$

where σ is the surface tension, $\Delta\rho = \rho_l - \rho_g$ is the density difference, g is the gravitational acceleration, Ja is the Jakob number, and Bo is the boiling number.

FLUENT only calculates bubble diameter using a simple relationship between bubble diameter and void fraction.

$$d_g = d_{g,o} \left(\frac{\alpha_g}{\alpha_{g,o}} \right)^{\frac{1}{3}} \tag{8}$$

where $d_{g,o}$ is the initial diameter and $\alpha_{g,o}$ is the volume fraction calculated without mass transfer.

RELAP code

The RELAP code is based on a one-dimensional, transient analysis code. It employs a non-homogeneous and non-equilibrium flow model for two-phase regions to predict pressures, temperatures, void fractions and flow rates. The code uses a six-equation formulation to handle the phasic continuity, momentum and energy conservation equations for each phase.

Void Fraction Comparison

The experimental data of Zeitoun and Shoukri [6] was used as the basis for comparison of the FLUENT, CFX and RELAP codes. The experimental data was taken from the measured void distribution in a vertical concentric annular tube flow. A 340 mm long heater was mounted on the 12.7 mm outside diameter of the inner tube. The 25.4 mm inside diameter outer tube was insulated. Subcooled water was pumped from below and the void fraction distribution was measured within the heated section. A total of ten test conditions of different subcooled temperatures θ_{in}, water flow rates, amounts of heat input q, and operational pressures P, were investigated. The CFD simulation cases were setup at the same conditions as in the experiment. The void fraction distribution along the heated section is presented in figures 2 to 11. The void fraction distribution is also compared with the study of Anglart and Nylund [1]. Only three cases were run using RELAP 5 as shown in figures 5, 6, and 10 which fall into the three ranges of Reynolds number shown in table 1.

For the low Reynolds number range, all three simulation codes used here predicted the trend of void fraction correctly. FLUENT and CFX resulted in a better prediction than both RELAP and the results of Anglart and Nylund [1]. Both RELAP and Anglart and Nylund [1] under-predicted the void fraction along the heated section. CFX under-predicts the void fraction where the working fluid is still in a highly subcooled state. After that CFX predicts the variation of void fraction in the moderate and slightly subcooled regions very well.

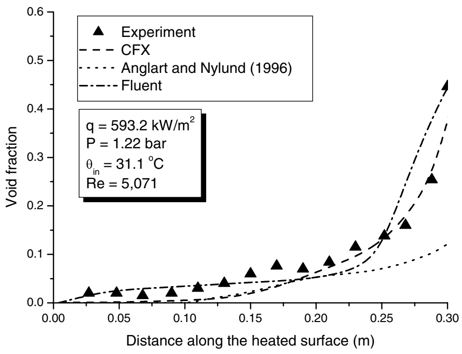

Figure 2: Case B1

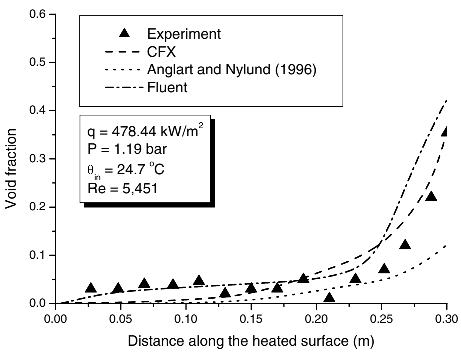

Figure 3: Case B2

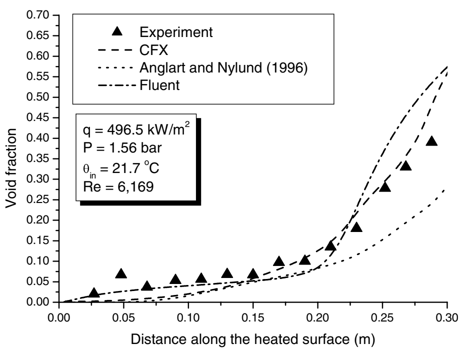

Figure 4: Case B3

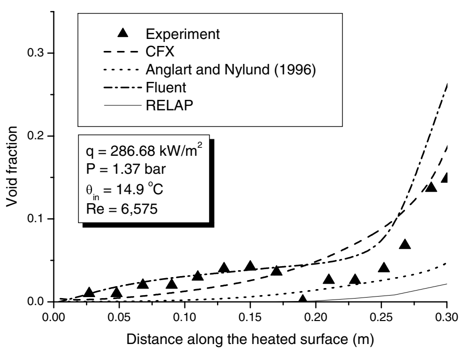

Figure 5: Case B4

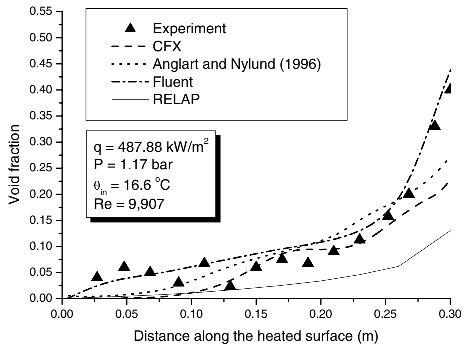

Figure 6: Case B5

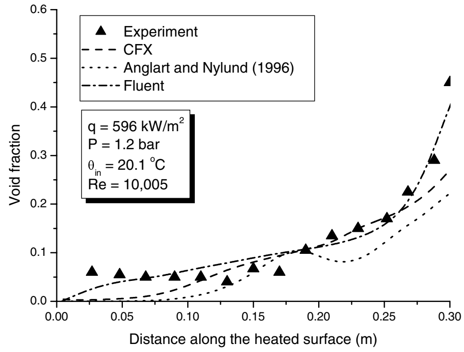

Figure 7: Case B6

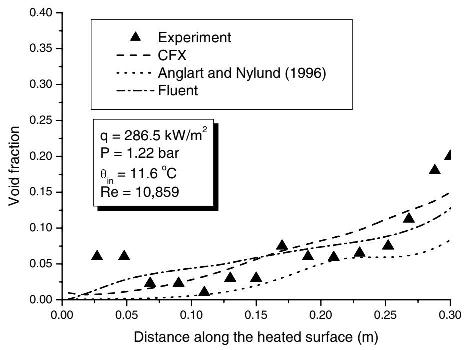

Figure 8: Case B7

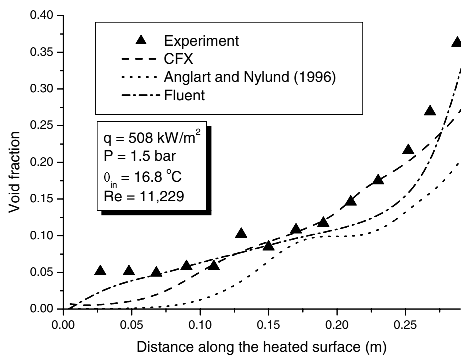

Figure 9: Case B8

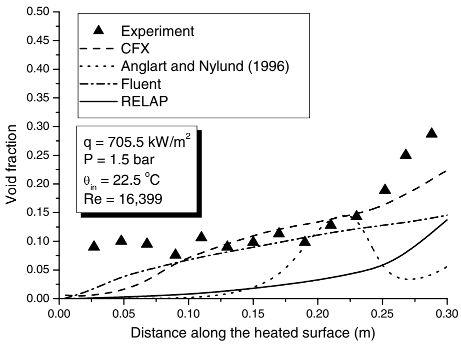

Figure 10: Case B9

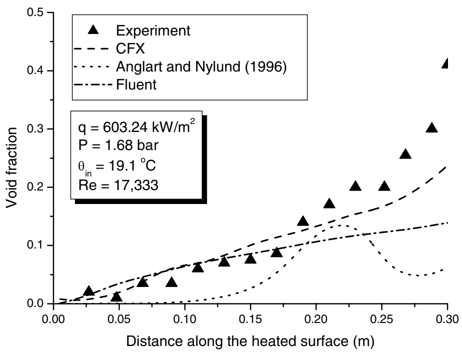

Figure 11: Case B10

For the medium Reynolds number range, FLUENT results offer the best void fraction prediction in almost every case of this Reynolds number range. CFX prediction is very good in the moderate and slightly subcooled region. The RELAP result is lower than the experimental data for the whole distance along the heated section.

For the high Reynolds number range, most codes did not predict the void fraction very well. All codes under-predict the void fraction result for every region shown in figure 1. Of the three codes, CFX gives relatively the best agreement with the experimental result. However, under-prediction in the high subcooled region still exists.

Concluding Remarks

A CFD simulation of subcooled flow boiling in an concentric annular tube, which is commonly used in the nuclear reactor fuel elements, was studied. Two general purpose CFD packages, FLUENT 4.5 and CFX4-2, were validated with experimental data and also compared with the specially designed 1D CFD code for boiling simulation in nuclear reactors, RELAP. Modifications of adjustable parameters of evaporation and condensation equations were made in FLUENT. The bubble diameter calculation of CFX was modified by using an empirical correlation. The computational result of void fraction was plotted against the experimental result of Zeitoun and Shoukri [6].

The results showed that the general purpose commercial CFD codes perform quite well for all cases conducted in this study. FLUENT tends to give better prediction in the highly subcooled region, while CFX perform quite well to regions where the void fraction very high. RELAP predicted poorly for this vertical annular case. With the modification of mean bubble diameter, CFX predicted the void fraction relatively better than FLUENT and RELAP in the slightly subcooled region.

Table 1: Ten simulation cases

Reynolds Number Range	Case	Reynolds Number
Low	B1	5071
	B2	5451
	B3	6169
	B4	6571
Medium	B5	9907
	B6	10005
	B7	10859
	B8	11229
High	B9	16399
	B10	17333

Acknowledgements

This work was supported by the Australian Institute of Nuclear Science and Engineering (AINSE) as a part of AINSE Project 01/038.

References

[1] Anglart, H. and Nylund, O., CFD application to prediction of void distribution in two-phase bubbly flows in rod bundles, *Nuclear Engineering and Design*, **163**, 1996, 81– 98.

[2] Kristiansen, H., Fallet, T. and Bjornneklett, A., A study on the evaporation heat transfer in the cooling of high power electronics, in *The Tenth Annual IEEE Semiconductor Thermal Masurement and Management Symposium*, San Jose, California, 1994, 114– 120.

[3] Stephens, D. W. and Harris, J. A., Numerical modelling of two-phase flow a benchmark solution, in *The Thirteen Australasian Fluid Mechanics Conference*, Monash University, Melbourne, Australia, 1998, volume 2, 1005– 1008.

[4] Tu, J. Y., The influence of bubble size on void fraction distribution in subcooled flow boiling at low pressure, *International Communications in Heat and Mass Transfer*, **26**, 1999, 607– 616.

[5] Zeitoun, O. and Shoukri, M., Bubble behaviour and mean diameter in subcooled flow boiling, *Journal of Heat Transfer*, **118**, 1996, 110– 116.

[6] Zeitoun, O. and Shoukri, M., Axial voil fraction profile in low pressure subcooled flow boiling, *International Journal of Heat and Mass Transfer*, **40**, 1997, 869– 879.

[7] Zhao, Y., Masuoka, T., Ozoe, H. and Tsuruta, T., Analytical study on mechanism of boiling heat transfer in a moderate narrow space, in *Transport Phenomena in Thermal Science and Process Engineering*, Kyoto, Japan, 1997, 161– 166.