Numerical simulation of flow boiling of cryogenic liquids in tubes

Li X.D., Wang R.S.*, Gu A.Z.

Institute of Refrigeration and Cryogenics, Shanghai Jiaotong University, Shanghai, 200030, China

The two-fluid model of two-phase flow, which is included in a commercial CFD code, was used to numerically predict the boiling flows of cryogenic liquid in a vertical tube. Important information, including heat transfer coefficient, void fraction and pressure loss was obtained. The numerical results were in satisfactory agreement with the experimental data available in the literature, which indicated that CFD offers an economical and effective method for cryogenic engineering designing and studies.

INTRODUCTION

Flow boiling is widely encountered in cryogenic systems. Accurate knowledge about cryogenic two-phase flows is very important to the design and optimization of these systems. Experimental studies are very costly and, furthermore, are subject to many limitations. On the contrary, CFD (Computational Fluid Dynamics) simulations may complement plenty of useful information without the considerable expense of experimental facilities. In recent years, CFD is increasingly utilized in studies of cryogenic systems, such as the jobs by Ishimoto [1] and Boukeffa [2]. However, there is no any report on the successful application of CFD in the flow boiling of cryogens. To overcome this, a two-fluid model of two-phase flows, which is included in a commercial CFD code CFX-4 (AEA Technology), was utilized to numerically predict the boiling flows of liquid nitrogen in a vertical tube. The numerical results were in an encouraging agreement with the experimental data available in the literature.

INTERPHASE TRANSFER MODELS

In a two-fluid model, two sets of conservation equations governing the balance of mass, momentum and energy of each phase are solved. Sine the macroscopic fields of one phase are not independent of the other phase, the interaction terms which couple the transport of mass, momentum and energy across the interfaces appear in the field equations.

Inter-phase momentum transfer

Previous studies [3] have indicated that unless inter-phase momentum transfer terms are accurately modeled, the advantage of the two-fluid model over other two-phase flow models disappears and, numerical instabilities result. The inter-phase momentum transfer is usually modeled with the interfacial forces, which include the drag force and the “non-drag” forces. All of these forces are modeled using the correlations recommended by CFX-4 [4].

Inter-phase heat transfer

Heat transfer across a phase boundary is usually described in terms of an inter-phase heat transfer coefficient, and the heat transfer coefficient is usually expressed in terms of a dimensionless Nusselt number, which can be calculated using the Ranz-Mashall [5] correlation.

<u>Inter-phase mass transfer</u>

The mass transfer between phases could be modeled using the RPI boiling model. In the model, the total wall heat flux is split into three parts: heat transfer rates due to convection, due to quenching and to evaporation, among which, the last one determines the mass transfer rate to the vapor phase at the walls. In the interior of the flow, the mass transfer rate depends on the liquid temperature. When the liquid is subcooled, there is a bulk condensation, otherwise, evaporation occurs. In addition, in the issue investigated in the present job, the pressure loss is negligibly small relative to the system pressure, therefore, the saturated temperature and the latent heat of the liquid can be assumed as equal along the tube.

In order to describe the change of the bubble diameter, it is assumed that the bubble diameter varies linearly depending on the liquid temperature between two reference diameters calculated using the equation proposed by Zeitoun and Shoukri [6], as recommended by Tu [7].

NUMERICAL SIMULATION

Experiments in reference [8] were numerically investigated. In the experiments, subcooled liquid nitrogen flows upwards into an electrically heated vertical tube with a diameter of 10mm and a length of 1850mm. According to the geometry of the tube, a full-scale three-dimensional computational domain was built to perform the simulations, and the domain was dispersed into about 150000 meshes in the body-fit coordinate, as shown in Figure 1.

The inlet pressure was 0.7MPa and the inlet liquid subcooling was 7.8K. Other boundary conditions are shown in Table 1.

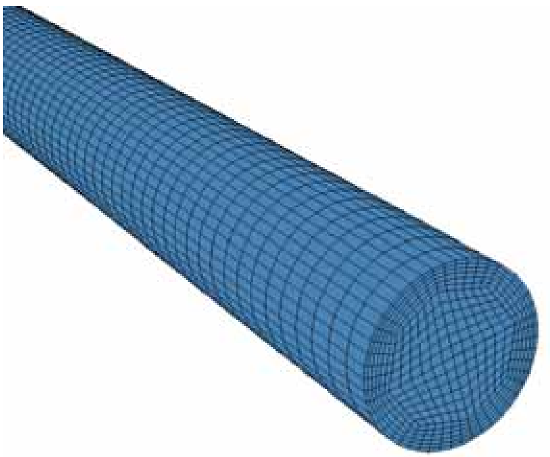

Figure 1 the computational domain and the meshes

Table 1 Boundary Conditions of the Simulations

Case No.	Mass flow rate [kg/(m^{2}s)]	Heat flux [W/m^2]
1	310	17530
2	330	20980
3	310	24710
4	300	29060
5	320	33120
6	320	37510

The two-fluid model was solved using the Inter-Phase Slip Algorithm (IPSA). The turbulence was modeled using the *k*-*ε* model and the bubble-induced turbulence in the liquid was modeled using the model proposed by Sato [9].

RESULTS AND DISCUSSION

Heat transfer coefficient

The circumference-averaged heat transfer coefficients along the tube length are given in Figure 2. Where, the solid lines are the simulated heat transfer coefficients and the marks stand for the experimental values. It can be seen that the simulated heat transfer coefficients at all heat fluxes investigated are in satisfactory agreement with the experimental data.

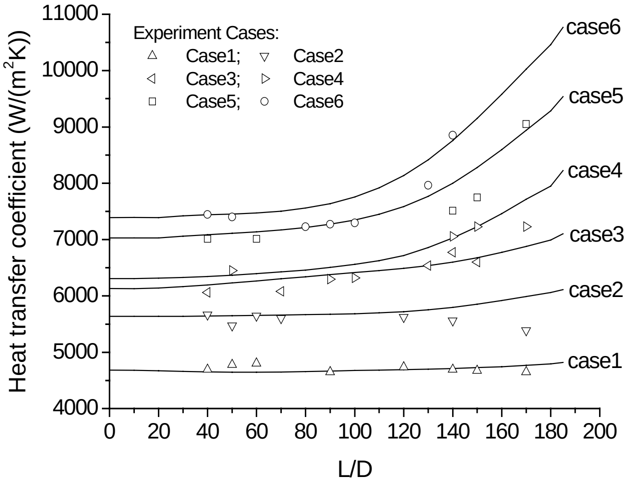

Figure 2 circumference-averaged heat transfer coefficient along the tube length

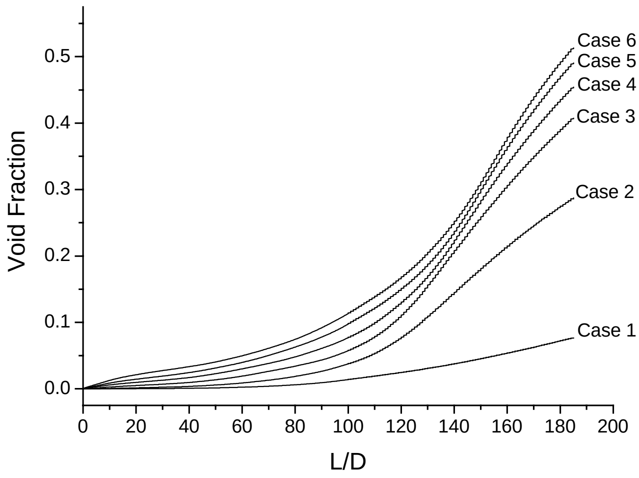

Figure 3 void fraction along the tube length

Void fraction

Area-averaged void fractions along the tube are shown in Figure3. Since in the experiments, the authors did not measure the void fraction and the pressure loss that will be discussed in the next section, the correctness of these parameters could be evaluated according to the heat transfer coefficients.

Take the related heat transfer coefficients in Figure 1 into consideration, it could be safely concluded that only when the void fraction is higher than a certain value could the heat transfer coefficient of a two-phase flow be obviously higher than that of its single-phase counterpart.

Pressure loss

The pressure losses of the two-phase flows are shown in Figure 4, in which, the maximum pressure loss is 3.2% of the inlet pressure. Under this value of pressure loss, the changes of the latent heat and the saturated temperature are negligible. Therefore, the hypothesis of constant physical properties is acceptable. It also can be found out that the pressure loss over the length decreases as the heat flux increases. This is due to the fact that the gravitational one is dominant among the three components of the pressure loss in the present issue. Since the density of the vapor is much smaller than that of the liquid, as the volume of vapor increases, the gravitational pressure loss gets smaller, which makes the total pressure loss decreases.

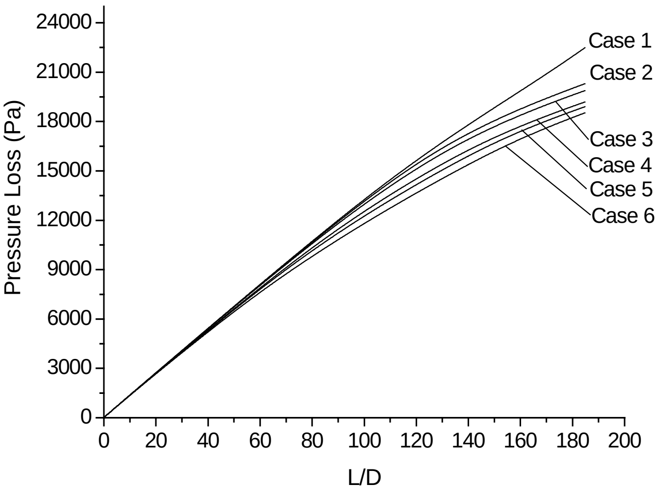

Figure 4 pressure loss along the tube length

CONCLUSION

Flow boiling of subcooled liquid nitrogen in a vertical tube was numerically simulated using a two-fluid model of two-phase flows, which is included in a commercial CFD package CFX-4. Comparison between the simulated heat transfer coefficients and the experimental values available in the literature showed an encouraging agreement. Since other parameters such as the void fraction and the pressure loss, which are not available in the related literature, are essentially related to the heat transfer coefficients, the validity of them therefore could be rationally explained according to that of the heat transfer coefficients.

It also can be concluded from the present job that CFD prediction can offer plenty of important information which is difficult to be measured in experiments, therefore it offers an effective and powerful method for cryogenic engineering designing and studies.

REFERENCES

1 Ishimoto, J. and Kamijo, K., Numerical simulation of cavitating flow of liquid helium in venturi channel, Cryogenics(2003), 43 9-17

2 Boukeffa, D. and Boumaza, M., Experimental and numerical analysis of heat loss in a liquid nitrogen cryostat, Applied Thermal Engineering(2001) 21 967-975

3 Ishii, M., Mishima, K., Two-fluid model and hydrodynamic constitute relations**.** Nuclear Engineering and Design 82, 1984:107-126

4 AEA Technology, CFX-4.3 User Guide Volume 3. AEAT, U.K., (1997)

5 Ranz,W.E. and Marshall, W.R., Chemical Engineering Progress(1952) 48 141-148

6 Zeitoun, O. and Shoukri, M., Bubble behavior and mean diameter in subcooled flow boiling, ASME Journal of Heat Transfer, (1996), 118 110-116

7 Tu, J.Y. and Yeoh, G.H., On numerical modelling of low-pressure subcooled boiling flows, International Journal of Heat and Mass Transfer(2002), 45 1197-1209

8 Klimenko, V.V. and Sudarchikov, A.M., Investigation of forced flow boiling of nitrogen in a long vertical tube, Cryogenics, (1983), 23 379-385

9 Sato, Y. and Sekoguchi, K., Liquid velocity distribution in two-phase bubble flow, International Journal of Multiphase Flow(1975), 2 79-95

* Corresponding author. Tel: +86-21-6293 2602
E-mail address: rswang@sjtu.edu.cn (R.S. Wang); leexiangdong@sjtu.edu.cn (X. D. Li)