Sensitivity Study for Wall Boiling Model in ANSYS CFX

Gong Hee Lee [a]*, Young Seok Bang [a], Sweng Woong Woo [a], Ae Ju Cheong [b]
[a]*Safety Analysis & Evaluation Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
[b]*Safety Issue Research Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
**Corresponding author: ghlee@kins.re.kr*

1. Introduction

Because boiling heat transfer was crucial for the analysis of operation and safety of both nuclear reactors and conventional thermal power systems, extensive studies have been made to develop a variety of methods either to evaluate the boiling heat transfer coefficient or to assess the onset of critical heat flux (CHF) at various operating conditions of heating channels [1,2].

Because of the limitation in grid resolution for a CFD simulation in comparison with the microscopic length scales of the wall boiling process, empirical closure for some underlying physical process is needed.

The main objective of the present study is to conduct the sensitivity study for wall boiling related models in ANSYS CFX R.14 [3] in order to examine the effect of model components on wall boiling heat transfer in an axis-symmetric vertical heated pipe.

2. Analysis Model

2.1 Test Facility and Test Conditions

Bartolomei and Chanturiya [4] performed an experiment on a subcooled convective flow boiling in an axis-symmetric vertical heated pipe. Test facility included upward boiling water flow in a 2 m long heated pipe of 15.4 mm diameter. Pressure in a pipe was maintained at 4.5 MPa. Subcooling at inlet was 58.2 K and the mass flux was 900 kg/m^{2}s. A uniform heat flux of 0.57 MW/m^2 was applied along a pipe wall. Axial distributions of bulk liquid temperature and bulk void fraction were measured.

2.2 Wall Boiling Model

A mechanistic model for the wall boiling heat transfer commonly used in CFD two-phase flow simulations is based on a wall heat flux partitioning principle where the total heat flux is distributed to the liquid and vapor phases via the following mechanisms (see Fig. 1):

- *Evaporation heat transfer (Q_e)* to the vapor phase, during bubble growth, through the liquid-micro layer located beneath the nucleating bubble.
- *Quenching heat transfer (Q_q)* to the liquid phase occurring after bubble departure within the wall area of nucleating bubble influence.
- *Single-phase turbulent heat transfer (Q_c)* to the liquid phase in the wall area outside the area of bubble influence.

In this study, RPI (Rensselaer Polytechnic Institute) model [3] in CFX R.14 was used.

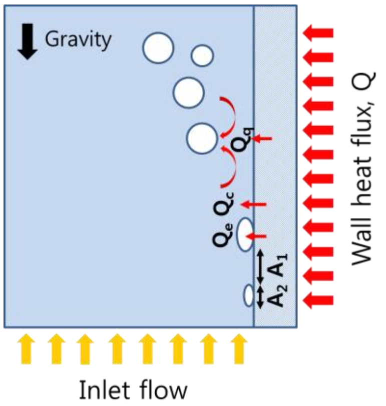


Fig. 1. Wall heat flux partitioning

3. Numerical Modeling

3.1 Numerical Method

The flow was assumed to be steady and turbulent. High resolution scheme was used for the convection terms of momentum equations and 1st order upwind scheme was used for the convection terms of turbulence equations. The solution was considered to be converged when the residuals of variables were below 10^{-6} and the variations of the target variables were small. The effect of buoyancy force was modeled with density difference model. The generation of turbulent kinetic energy due to buoyancy was not considered in k and ε equation. Simulation was conducted with the commercial CFD software, ANSYS CFX R.14 [3].

3.2 Turbulence Model

Standard k-ε model, which is one of Reynolds-averaged Navier-Stokes (RANS)-based two equation turbulence models, was used. It is well known that this model has been widely used in the various industrial applications and has a superior convergence in comparison with other turbulence models.

3.3 Grid System and Boundary Conditions

A hybrid (hexahedra and wedges) type grid system was used for the computational domain that had the same dimension as the test facility. Total number of nodes was 6191.

Mass flux of 900 kg/m^{2}s was imposed at normal to inlet boundary. Turbulence intensity at inlet was assumed to be 5.0%. Average pressure over whole outlet option with the relative pressure of 0 Pa was used as an outlet boundary condition. No-slip condition was applied on the solid wall. A uniform heat flux of 570,000 W/m^2 was applied along the pipe wall. Axis-symmetric boundary condition was applied to save the computation time.

4. Results and Discussion

4.1 Effect of Wall Area Fraction

The heat flux partitioning model separates the whole wall surface into two fractions (see Fig. 1):

· Fraction A_2 is influenced by the vapor bubbles, formed on the wall.

· Fraction A_1 is the rest of the wall surface and represents the part of the wall surface that does not experience the presence of the vapor phase.

The area fraction values A_1 and A_2 play an important role in the heat flux partitioning model. They are related to the nucleation site density per unit wall area and to the influence area of a single bubble forming at the wall nucleation site.

Fig. 2 shows the effect of wall area fraction (A_2) on both liquid temperature and void fraction in the axial direction. As the magnitude of A_2 increased liquid temperature in the axial direction came close to the experimental data. A smaller magnitude of A_2 might give more heat transfer to the liquid and therefore liquid temperature in the axial direction increased.

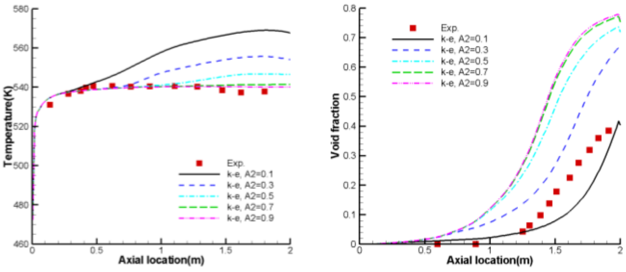
Meanwhile, as the magnitude of A_2 decreased void fraction in the axial direction came close to the experimental data. A smaller magnitude of A_2 might give less heat transfer to the vapor bubbles and therefore void fraction had smaller magnitudes.

As a result, there was no optimal value of A_2 to predict accurately both liquid temperature and void fraction in the axial direction.

4.2 Effect of Lift Force Model

In ANSYS CFX R.14 [3], there are three types of lift force models. In this study, Tomiyama lift force model was selected to examine the effect of lift force model on both liquid temperature and void fraction in the axial direction. This model is applicable to the lift force on larger-scale deformable bubbles in the ellipsoidal and spherical cap regimes [3].

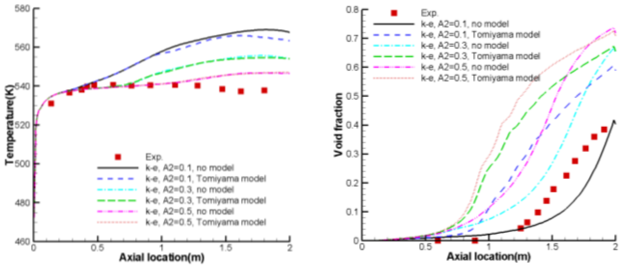
Fig. 3 shows the effect of wall area fraction (A_2) and lift force model on both liquid temperature and void fraction in the axial direction. Tomiyama lift force model predicted the lower liquid temperature in comparison with the no lift force model case. As the magnitude of A_2 increased the difference in liquid temperature for both case decreased.



(a) (b)

Fig. 2. Effect of wall area fraction A_2 on (a) liquid temperature (b) void fraction in the axial direction.

Meanwhile, there were significant differences in void fraction in the axial direction depending on the lift force model. In most cases, Tomiyama lift force model predicted the higher void fraction in comparison with the no lift force model case. As the magnitude of A_2 increased the difference in void fraction for both case decreased. As a result, Tomiyama lift force model did not show the improved prediction.



(a) (b)

Fig. 3. Effect of wall area fraction A_2 and lift force model on (a) wall temperature (b) void fraction in the axial direction.

Acknowledgments

This study was conducted under the financial support of the Nuclear Safety and Security Commission of Korea [project title: Development of Regulatory Evaluation Technologies for Thermal-hydraulic Safety]. The authors gratefully thank Mr. Lee Yong Gap in ANFLUX Inc. for the valuable technical comments.

REFERENCES

[1] M. Z. Podowski, R. M. Podowski, Mechanistic Multidimensional Modeling of Forced Convection Boiling Heat Transfer, Science and Technology of Nuclear Installations, Vol.2009, p.1, 2009.

[2] J.-M. Le Corre, S.-C. Yao, C. H. Amon, A Mechanistic Model of Critical Heat Flux under Subcooled Flow Boiling Conditions for Application to One- and Three-dimensional Computer Codes, Nuclear Engineering and Design, Vol.240, p.235, 2010.

[3] ANSYS CFX, Release 14.0, ANSYS Inc.

[4] G. G. Bartolomei, V. M. Chanturiya, Experimental Study of True Void Fraction when Boiling Subcooled Water in Vertical Tubes, Thermal Engineering, Vol.14, p.123, 1967.