International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 - 5161

Available at http://inpressco.com/category/ijcet

Research Article

Modeling of Vapor Bubble Condensation in Vertical Rectangular Channel using Open FOAM

Ebin P S[Å*], Harikrishnan G[Å] and Jayakumar J S[Å]

[Å]Department of Mechanical Engineering, Amrita Vishwa Vidyapeetham University, Amritapuri, Kerala, India

Accepted 16 January 2014, Available online 01 February 2014, **Special Issue-2, (February 2014)**

Abstract

This paper deals with the development and validation of a new solver in Open FOAM, bubble Condense Foam for modeling a condensing vapor bubble in subcooled boiling flows. This solver is used to predict the behavior of a vapor bubble in the flow in terms of bubble diameter and void fraction with respect to parameters such as subcooling temperatures, initial bubble diameter etc. VOF model is used to predict the interface of the bubble. Three conservation equations each for liquid and the gas phase are solved. Standard k-ε turbulence model consisting of two transport equations for turbulent kinetic energy and turbulent dissipation are used. The code developed is found to be capable of calculating condensation. However previous studies for modeling bubble condensation doesn't consider the lift-drag models, this study incorporates the lift-drag models with governing equations, which improves significantly the bubble diameter prediction.

Keywords: *Condensation, VOF, Open FOAM, CFD, Numerical, bubble Foam, bubble Condense Foam, lift-drag models.*

1. Introduction

In the subcooled flow boiling, bubble condensation is a key factor to describe the heat and mass transfer phenomena and it is classical part of micro heat transfer. It is encountered in many industrial applications, especially electronics cooling and nuclear reactor. In bubble condensing process, bubble size, shape and void fraction will change continuously; meanwhile, they have significant influence on heat and mass transfer through the bubble interface transportation. Therefore, in order to understand the subcooled flow boiling, it is a challenge to acquire knowledge on condensing bubbles behavior.

To account for the wall nucleation and condensation in the subcooled boiling regime, a model was developed, MUSIG in CFX4.4 (Tu and Yeoh, 2003). This model predict the bubble diameter, void fraction and gas liquid velocities for different mass and heat fluxes and inlet subcooling temperatures. However, significant weakness of the model is still in the prediction of the vapor velocity. But they claimed that deficiency of the model can be overcome by the consideration of algebraic slip model to account for the bubble separation.

A mechanistic model has been developed for the heat flux partitioning during subcooled flow boiling (Basu *et al.*, 2005). The basis of this is that the entire energy from the wall is first transferred to the superheated liquid layer adjacent to the wall. A fraction of this energy is then utilized for sensible heating of the bulk liquid. The contribution of each of the mechanisms for transfer of heat to liquid-forced convection and transient conduction, as well as the energy transport associated with vapor generation has been quantified in terms of nucleation site densities, bubble departure and lift-off diameters, bubble release frequency, flow parameters like velocity, inlet subcooling, wall superheat, and fluid and surface properties including system pressure. The model developed shows that transient conduction component can become dominant mode of heat transfer at very high super heat. Hence, velocity does not have much effect at high superheats. This is particularly true when boiling approaches fully developed nucleate boiling. Also the model developed allows prediction of the wall superheat as a function of applied heat flux or axial distance along the flow direction

Time development of subcooled boiling was simulated and effects of evaporation, condensation, vapor-phase convection, vapor-phase diffusion on the flow field and pressure drop were analyzed byIvo *et al*. (2006). The bubble volume, generated due to evaporation process, is filled with liquid and included as a source term in the continuity equation for the liquid phase. Thus, the single phase form of transport equations is preserved and bubbles were retained in the boundary layer near the heated surface. Heat transfer phenomena during subcooled boiling flow affect the shape and area of the varying interface significantly (Jeon *et al*, 2009). Heat transfer characteristics of the subcooled pool boiling, its mechanism, and the development of boiling and condensation model were done by Ose and Kunugi, (2011) using a solver, MARS (Multi-interface Advection and

*Corresponding author: **Ebin P S**

Reconstruction Solver) developed by them.

A study was conducted for investigating the parametric dependency of bubble life time (Pan et al 2012). With the increasing of subcooling, the bubble life time reduces significantly and is proportional to system pressure. It was also noted that the size of the channel influence the bubble deformation behavior significantly. To investigate the fluid flow around bubbles attached to heated walls, a three dimensional CFD model is developed in OpenFOAM. The pressure-velocity coupling has been treated using PISO (Pressure Implicit Splitting of Operators) procedure. The equations have been discretized by the QUICK interpolation. The transient solutions of the governing equations in a domain containing the bubbles and the surrounding liquid have been obtained. The nucleation, growing and detachment processes have been analyzed. Special attention has been given to the bubble detachment diameter. It has been found that bubble detachment diameter depends on the contact angle, operating pressure and properties of the fluid (Lames *et al*, 2012). Even though this model correctly predicts the behavior of attached bubbles, become insufficient to predict the bubble behavior far away from the heated walls.

In the present study, a condensing vapor bubble in subcooled boiling flow within a vertical rectangular channel has been modeled in OpenFOAM using VOF multiphase flow model. For modeling bubble condensation bubbleFoam is taken as the base solvers. The solver is modified by adding routines to solve energy equation and to model phase change during condensation and interfacial forces. Validation of the solver was carried out using the results presented in previous work (Jeon *et al,* 2009). Subsequently analysis was done incorporating lift and drag models.

2. Description of problem

Subcooled boiling flow is characterized by the growth of vapor bubbles at a heated solid surface and their subsequent condensation or collapse inside the subcooled liquid. These bubbles originate at cavities, pits and scratches on the heater surface where vapor or non-condensable gases are trapped. When the temperature at the liquid solid interface exceeds the saturation temperature by a few degrees, nucleation sites become active and boiling commences. An active nucleation site produces vapor bubbles which go through a typical periodic cycle of nucleation, growth, departure and collapse (condensation) followed by a waiting period. Figure.1 shows the condensation of vapor bubble which is formed at the solid is occurred when the local fluid temperature come below the saturation temperature. Behavior of such a condensing vapor bobble is investigated in this work.

3. Outline of the work

In subcooled boiling flow, bubble condensation is the key parameter to describe heat and mass transfer phenomena. It significantly affects the shape and area of the varying interface thus the behavior of the condensing bubble becomes different from that of adiabatic bubble. Therefore, in order to understand the bubble behavior in subcooled boiling flow, a numerical study of condensing bubbles considering heat and mass transfer through the bubble interface is required.

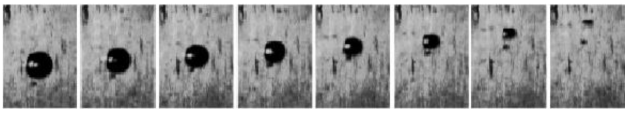


Figure.1 Bubble collapse in subcooled boiling flow (Pan et al, 2012)

The present study focused on how to simulate bubble condensation with a CFD code. Moreover, using the VOF model, the behavior of condensing bubbles in subcooled boiling flow was investigated. In order to simulate the condensing bubble with OpenFOAM, basic solver, bubbleFoam have to be modified. The results of CFD simulation were compared with the literature data. The VOF model is utilized to track the interface. The fundamental behavior of condensing bubble in terms of bubble diameter variation is investigated.

3.1 Development of solver

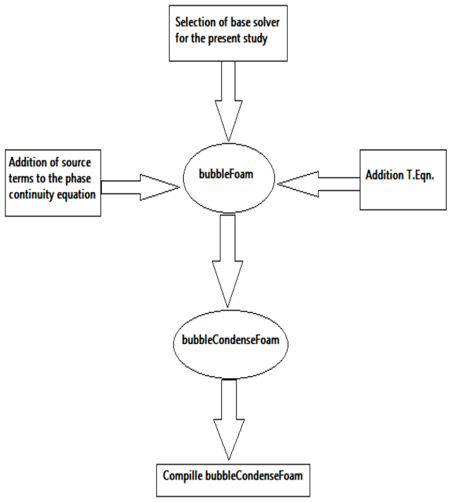


Figure.2 Development of bubbleCondenseFoam

3.2 case structure for the solver

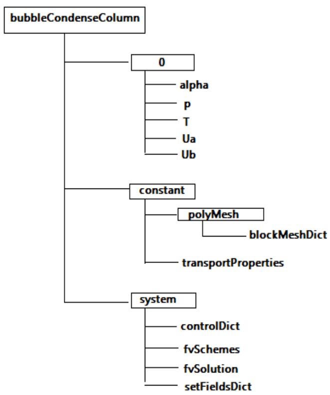


Figure.3 case directory structure

3.3 bubble Condense Foam working loop

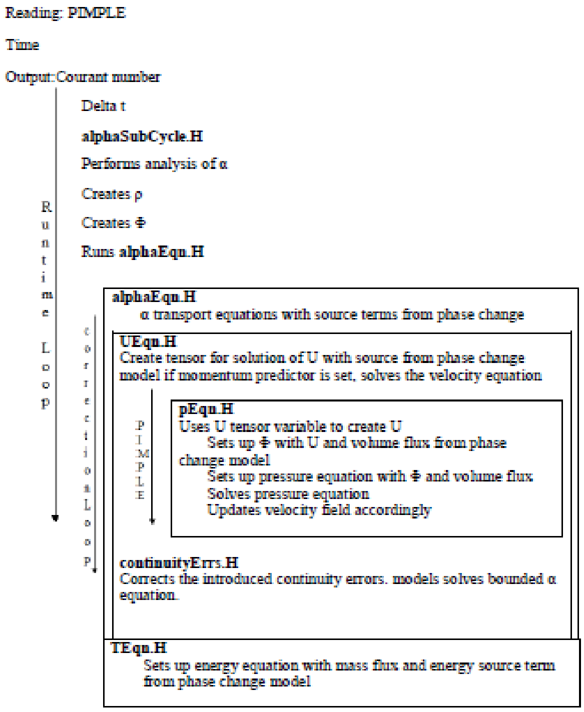


Figure.4 bubbleCondenseFoam working loop

4. Numerical Methodology

4.1. Interface tracking with VOF

The solver uses a VOF approach to track the interface and with this information the momentum exchange between the phases is derived. The VOF uses a scheme that tracks the interface of the dispersed phase by utilizing the phase fraction within the cell (Hirt and Nicholas, 1981). Every cell stores a value between 0 and 1, which denote empty and full of the continuous phase, respectively. This function α depends on time and position, meaning α = α(t,x), can be partially differentiated and moves with the fluid, as can be seen in equation (1).

$$\frac{\partial}{\partial t}(\alpha_k\rho_k) + \nabla(\alpha_k\rho_k U_k) = \dot{m} \qquad (1)$$

where $\dot{m}$ is mass source term and α_k represents the phase fraction of phase k. In this study, this mass source term is modeled to simulate mass transfer between the phases during condensation. Properties values at each cell were determined based on void fraction values as in equation (2).

$$\rho = \alpha_k\rho_k + (1-\alpha_k)\rho_l \qquad (2)$$

The phase momentum equation is

$$\frac{\partial}{\partial t}(\alpha_k\rho_k U_k) + \nabla.(\alpha_k\rho_k U_k U_k) + \nabla.(\alpha_k\tau_k) + \nabla.(\alpha_k R_k) = -\alpha_k\nabla p + \alpha_k\rho_k g + M_k \qquad (3)$$

In which τ_k is the phase laminar stress tensor, R_k is the phase Reynolds stress tensor, p is the pressure, g is the gravitational acceleration vector, and M_k is the momentum exchange term. The laminar stress tensor is defined, for each phase, as

$$\tau_k = -\rho_k v_k[\nabla \mathrm{U_k} + \nabla^{\mathrm{T}}\mathrm{U_k}] + \tfrac{2}{3}\rho_k v_k(\nabla.\mathrm{U_k})\mathrm{I} \qquad (4)$$

where v_k is the molecular kinematic viscosity of the fluid constituting phase *k*. *I* is the identity matrix. The phase Reynolds stress tensor is given by

$$R_k = -\rho_k v_{k,t}[\nabla \mathrm{U_k} + \nabla^{\mathrm{T}}\mathrm{U_k}] + \tfrac{2}{3}\rho_k v_{k,t}(\nabla.\mathrm{U_k})\mathrm{I} + \tfrac{2}{3}\rho_k k_k I \qquad (5)$$

where k_k is the phase turbulent kinetic energy, and is the turbulent kinematic viscosity. The momentum exchange terms can be decomposed in a drag contribution and a virtual mass contribution.

$$M_k = M_{k,drag} + M_{k,lift} + M_{k,vm} \qquad (6)$$

The newly added energy equation for modeling bubble condensation phenomena is

$$\frac{\partial}{\partial t}(\rho C_p T) + \nabla(\vec{v}\,\rho C_p T) = \nabla(k\,\nabla T) + (\dot{m}.h_{fg}) \qquad (7)$$

4.2. Modeling of bubble condensation

The source terms in the governing equations should be modeled to simulate bubble condensation because the default bubbleFoam solver cannot simulate heat and mass transfer through the bubble interface.

4.2.1. Modeling of interfacial heat transfer coefficient

In order to model the bubble condensation, this study uses equation (8) for the convective heat and mass transfer

$$q = h_i A_b(T_{sat} - T_l) = \dot{m}.h_{fg} \qquad (8)$$

where h_i is the interfacial heat transfer coefficient, T_{sat} and T_l are the vapor and liquid temperatures, respectively, A_b is the interfacial area of the bubble, *m* is the total mass transfer rate and h_{fg}. In order to get interfacial heat transfer coefficient, this study uses some correlations which is obtained from Hansen (2009).

$$Pr_b = \frac{\rho_b v_b C_{pb}}{k_b} \qquad (9)$$

Bubble Reynolds number can be calculated as,

$$Re_b = \frac{V_{rel} D_b}{v_b} \qquad (10)$$

$$V_{rel} = |U_b - U_l| \qquad (11)$$

$$D_b = 6.\alpha.D_0 \qquad (12)$$

U_b and U_l are the bubble and fluid velocity; D_b and D_0 are the instantaneous and initial diameter of the bubble respectively.

$$Nu = 2 + \left(0.6\ Re_b^{\frac{1}{2}} Pr_b^{\frac{1}{3}}\right) \qquad (13)$$

The interfacial heat transfer,

$$h_i = \frac{Nu\,.k_b}{D_b} \qquad (14)$$

4.2.2 Modeling of source terms

The source terms are modeled using h_i, calculated from equation (14). The total mass transfer from vapor to liquid is obtained as

$$\dot{m} = \frac{\dot{q}}{h_{fg}} = \frac{h_i A_b (T_{sat} - T_l)}{h_{fg}} \qquad (15)$$

where h_{fg} is the latent heat and A_b is the bubble surface area.

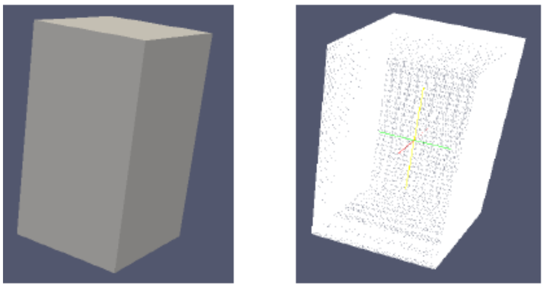
$$A_b = \pi . {D_b}^2 \qquad (16)$$

The energy transfer rate, or the energy source term in the energy equation can be modeled as

$$\dot{m}. h_{fg} = h_i (T_{sat} - T_l) A_b \qquad (17)$$

5. Simulation procedure

(a) The geometry and meshes are generated in a three-dimensional Cartesian coordinate system using ***blockMeshDict*** file is shown in figure. 5 (a) and (b). The dimensions of the created geometry are 15 x 15 x 30 mm^3 and the flow direction is upward.



(a) Geometry (b) Mesh

Figure.5 Geometry and mesh for the problem

(b) The initial and boundary conditions such as liquid temperature and liquid velocity are set in the ***0*** directory. Bubble is created in the flow field using ***setFieldsDict*** file, which is shown in figure.6.

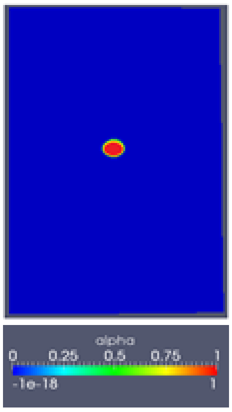


Figure.6 Void fraction for bubble with Do = 1mm at t=0s

c) Discretisation schemes and solver settings such as tolerances are specified in the ***fvSchemes*** and ***fvSolution*** files located in ***system*** directory.

d) Fluid properties such as density, specific heats etc. are given in ***transportProperties*** file located in ***constant*** directory.

e) Start time, end time, time interval and write interval are specified in the ***controlDict*** file in ***system*** directory.

f) Run the created solver ***bubbleCondenseFoam*** and view the results in ***paraView.***

5. Results and discussions

The bubble shape from initial spherical shape is changed to elliptical and then diffused into fluid stream as time proceeds. Table.6.1 also shows the comparison of OpenFOAM results with experimental results of Seoul National University (SNU) and the FLUENT work done by Jeon et al (2009).

Table 1 Comparison of OpenFOAM simulation result with experimental and FLUENT results for bubble with D_o = 1mm and ΔT_{sub} = 20K

TIME (S)	VARIATION OF *BUBBLE SHAPE* (FLUENT RESULTS) Jeon et al (2012)	VARIATION OF *VOID FRACTION* (OpenFOAM RESULTS) PRESENT STUDY	Experimental Results (SNU)
0.000			
0.005			
0.010			
0.015			
0.020			
0.025			
0.030			

The results in table.1 are reported for bubble with initial diameter 2mm and with subcooling of 20K. Initially the bubble has round shape then it changes to a distorted elliptical shape due to the heat transfer from the bubble core to fluid across the interface. Bubble gets condensed fully at 0.033 s. In the previous studies lift-drag models are not considered, while in this study lift-drag models are added to predict the bubble behavior correctly. Addition of these models accelerates the bubble collapse. Figure.8 clearly shows the bubble diameter variation with time for *bubbleCondenseFoam* with and without lift-drag models.

5.1 Temperature variation

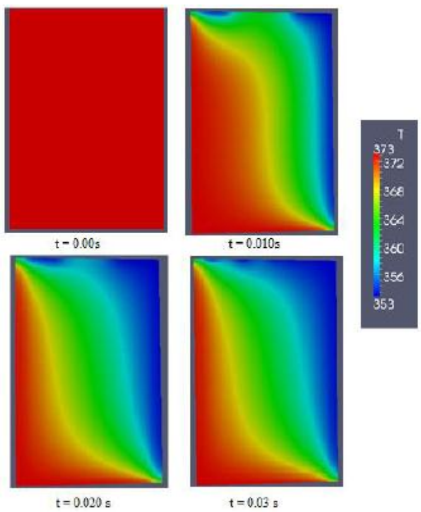


Figure.7 Temperature variation in computational domain during subcooling

Figure.7 shows the temperature distribution within the rectangular geometry during subcooling. The right wall is kept at lower temperature compared to left wall and internal field. The fluid within the channel starts to cool from right wall region to left wall region as time proceeds. The heat transfer from the fluid to the right wall will reduce the fluid temperature below saturation value. Hence by viewing the temperature distribution we get an idea about the intensity of subcooling. For rapid cooling, temperature will change rapidly from right to left wall; this will accelerate the bubble condensation.

5.2 Variation of bubble diameter

Bubble diameter decreases gradually with time as condensation proceeds. Figure.8 shows the variation of bubble diameter against time for bubble with initial diameter 4mm and subcooling of 10K. Results of present study show good agreement with the work of Jeon et al (2011). The reason for reduction in bubble diameter with time is that the heat transfer across the bubble interface from bubble inner core to subcooled liquid surrounding the bubble.

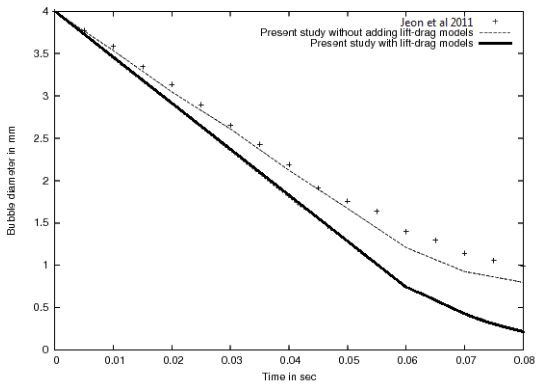


Figure.8 Variation of bubble diameter with time

5.2 Effect of subcooling on bubble condensation

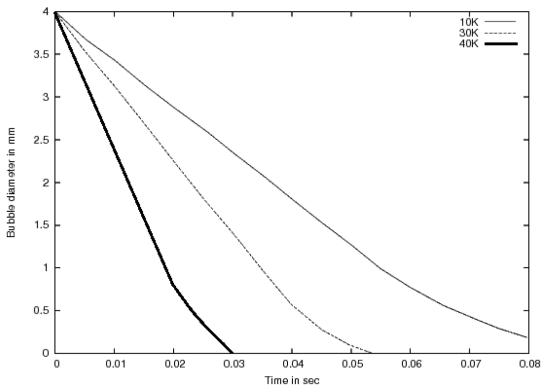


Figure.9 Bubble diameter variation with time for different subcooling temperatures.

As subcooling increases bubble condensation rate get accelerated due to the increased rate of heat transfer from bubble core to subcooled liquid. In figure. 9, bubble diameter variations are plotted for three different subcooling temperatures, 10K, 30K and 40K. From these figures it is clear that bubble get condensed early for 40K subcooling compared to other two subcooling temperatures.

5.3 Effect of initial diameter on bubble condensation

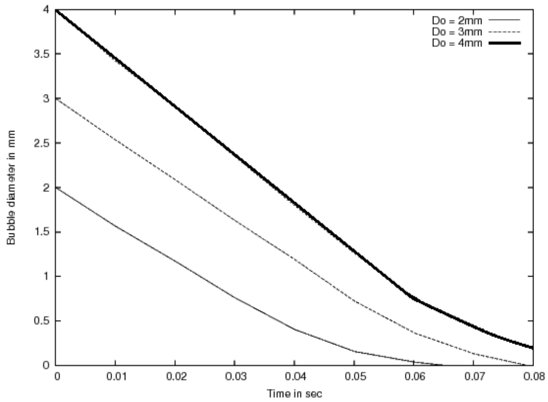


Figure.10 Effect of initial diameter of bubble on condensation

Bubble diameter variation for bubble with initial diameters 2mm, 3mm and 4mm is plotted with time for 10K subcooling is shown in figure.10 Bubble with 4mm diameter condenses slowly compared to bubbles with diameters 2mm and 3mm, they are condensing fully in 0.0646 s and 0.0784 s respectively. While 4mm bubble take beyond 0.08 s for full condensation.

5.4 Grid independency

For checking grid independency, bubble diameter variation is plotted for bubble with initial diameter 4mm and 10K subcooling for grids 25 x 75 x 25, 50 x 125 x 50 and 75 x 175 x 75. It was observed from figure.11 that results become stable for 75 x 175 x75 grids. Hence 75 x 175 x75 is selected as standard grid system for analyzing bubble condensation in subcooled boiling flow.

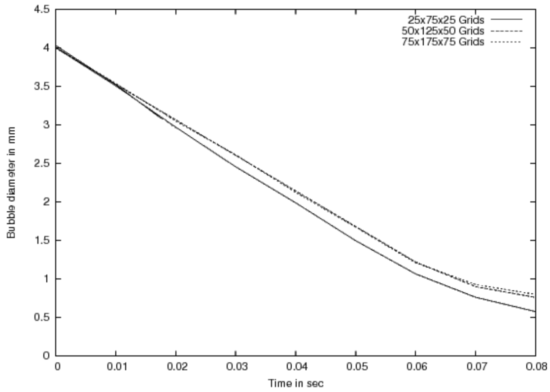


Figure.11 Test for grid independency.

Conclusions

1) As subcooling increases, condensation of vapor bubble accelerated due to the increase of heat transfer

from the bubble inner core to surrounding fluid across the bubble interface.

2) Bubble with large diameter takes more time to collapse than bubble with smaller diameter.
3) Addition of lift-drag models to the momentum equation accelerates the bubble condensation.
4) Lift-drag addition to previous work (Jeon *et al*, 2009) improves the bubble diameter prediction.

References

C.W Hirt, B.D Nichols, (1981), Volume of fluid (VOF) method for the dynamics of free boundaries, *Journal of computational physics*. Vol.39, pp. 201-225.

J Y. Tu.J.Y, G H .Yeoh, (2003), Development of a numerical model for subcooled boiling flow, *Third International Conference on CFD in the Minerals and Process Industries*, CSIRO, Melbourne, Australia.

Basu Nilanjana, Gopinath R. Warrier, Vijay K.Dhir, (2005), Wall Heat Flux Partitioning During Subcooled Flow Boiling, *Journal of Heat transfer*, Vol 27.

Ivo Kijenak, BostjanKoncar, BorutMavko, (2006), Modelling of subcooled Nucleate Boiling in a Vertical Channel with coupling of Bubble-Tracking and Two-Fluid Models, *Workshop on Modelling ad Measurements of two-phase Flow and Heat transfer in Nuclear Fuel Assemblies*. Sweden.

JeonSeong-Su, Kim Seong-Jin, Goon-Cherl Park, (2009), CFD Simulation of Condensing Vapor Bubble using VOF Model, World *Academy of Science, Engineering and Technology*, Vol. 36.

Hansen Rolf, (2009), Computational and experimental study of bubble size in bubble columns, *Ph.D. Thesis.*

OseYasuo , KunugiTomoaki, (2011), Development of A Boiling and Condensation Model on Subcooled Boiling Phenomena, *9th Eco-Energy and Materials Science and Engineering Symposium*, Vol 9, pp. 605 – 618.

Pan Liang-ming, Tan Zhi-wei, Chen De-qi, Xue Long-chang, (2012), Numerical investigation of vapor bubble condensation characteristics of subcooled flow boiling in vertical rectangular channel, *Nuclear Engineering and Design*. Vol 248, pp. 126– 136.