CFD MODELING OF SUBCOOLED FLOW BOILING FOR NUCLEAR ENGINEERING APPLICATIONS

B. Končar
“Jožef Stefan” Institute
Jamova 39,
1000 Ljubljana, Slovenia
Bostjan.Koncar@ijs.si

E. Krepper
Forschungszentrum Rossendorf e.V.
P.O.Box 510119, D-01314 Dresden, Germany
E.Krepper@fz-rossendorf.de

Y. Egorov
ANSYS Germany GmbH
Staudenfeldweg 12
D-83624 Otterfing, Germany
Yury.Egorov@ansys.com

ABSTRACT

In this work a general-purpose CFD code CFX-5 was used for simulations of subcooled flow boiling. The subcooled boiling model, available in a custom version of CFX-5, uses a special treatment of the wall boiling boundary, which assures the grid invariant solution. The simulation results have been validated against the published experimental data [1] of high-pressure flow boiling in a vertical pipe covering a wide range of conditions (relevant to the pressurized water reactor). In general, a good agreement with the experimental data has been achieved. To adequately predict the lateral distribution of two-phase flow parameters, the modelling of two-phase flow turbulence and non-drag forces under wall boiling conditions have been also investigated in the paper.

1 INTRODUCTION

Boiling and the associated heat transfer are some of the most important phenomena which not only affect the reactivity of the nuclear reactors but also determine the criticality of equipment in power generation facilities.

Subcooled boiling may occur in narrow flow passages between the heated fuel rods of a pressurized nuclear reactor core. Here vapour bubbles are generated in the micro cavities commonly designated as nucleation sites, which are randomly distributed over the heated surface. The vapour bubble is generated on a nucleation site when the surface temperature sufficiently exceeds the liquid saturation temperature at the local pressure. Bubbles may slide along the heated surface, eventually depart from it and migrate further into the subcooled liquid flow, where they are subjected to condensation. If the average temperature over the channel cross-section is lower than the liquid saturation temperature the boiling process is known as subcooled flow boiling. Subcooled flow boiling belongs to the category of complex two-phase flow phenomena associated with hydrodynamics, heat and mass transfer. Here, not only is vapour unevenly distributed over the channel cross-section, but, in addition, the distribution evolves along the flow, as both the void fraction and the width of the two-phase layer near the heated surface gradually increase. This non-homogeneous distribution of vapour significantly influences hydrodynamic and thermal processes, including mass transfer.

As the phenomena associated with subcooled flow boiling are typically three-dimensional, the multidimensional CFD modelling of such flow is necessary. Commonly a two-fluid Eulerian approach is used in CFD codes. Modelling of interfacial transfer...
determines the degree of thermal and hydrodynamic non-equilibrium between the phases, so the reliability and accuracy of the predicted results usually depend on the implemented constitutive relations for interfacial transfer. Due to the complexity of subcooled flow boiling, it is rather difficult to establish a general model able to cover a wide range of conditions.

In this work a general-purpose CFD code CFX-5 was used for simulations of subcooled flow boiling. Some modelling issues related to the modelling of turbulence and non-drag forces were investigated. The subcooled boiling model implemented in the test version of the CFX-5 code [2] is applied and validated in this work. The main improvements considering the wall boiling boundary conditions are briefly discussed. The model has been extensively validated against the published experimental data [1] of high-pressure flow boiling in a vertical pipe covering a wide range of conditions (relevant also to the pressurized water reactor). In this work, only the single-tube boiling experiments were simulated. Recently, the subcooled boiling flow calculations using CFX-5 have been performed also in real hot channels of a fuel assembly and were published in [3].

2 MATHEMATICAL MODEL

A multidimensional two-fluid Eulerian approach is used for mathematical description of subcooled boiling flow. The governing equations of the two-fluid model have been extensively described in many works (e.g. [4]), thus they will not be repeated here. The numerical method of the CFX-5 code is based on finite volume discretization on collocated grid arrangement. The liquid phase is dominant and is described as continuous while the vapour bubbles are described as a dispersed phase.

2.1 Turbulence modelling

Due to the lower density of vapour, it is commonly assumed that, in nucleate boiling flow, the motion of the dispersed vapour phase follows the fluctuations in the continuous liquid phase. Accordingly, the turbulence stresses are modelled only for the liquid phase, whereas the vapour phase is modelled by a simple zero equation model [5].

In the present study two different two-equation turbulence models for the liquid phase were used and compared. The first model is the k-ω based Shear Stress Transport (SST) model of Menter [5] and the other is the standard k–ε model, both belonging to the category of eddy viscosity turbulence models. In bubbly two-phase flows, an additional production of liquid turbulence generated by fluctuating wakes behind the large bubbles may occur. The so-called bubble-induced turbulence may be taken into account using two different approaches.

1. The model of Sato et al. [6] describes the bubble-induced turbulence by additional viscosity term, which is added to the molecular viscosity of the liquid phase \( \mu_l \) in the same way as the shear induced turbulence viscosity term \( \mu_l^{\text{turb}} \).

\[
\mu_l^{\text{eff}} = \mu_l + \mu_l^{\text{turb}} + \mu_l^b.
\] (1)

Here \( \mu_l^b \) represents the bubble-induced turbulence viscosity, which depends on the vapour phase volume fraction \( \alpha \), the local bubble diameter \( d_b \) and the relative velocity between the phases:

\[
\mu_l^b = C_{sb} \rho_l C_d d_b \left| \bar{u}_g - \bar{u}_l \right|.
\] (2)
Constant $C_{\mu b}$ commonly takes the value 0.6, as recommended by Sato et al. [6]. Besides the turbulence intensity, the bubble diameter $d_b$ also determines the interfacial momentum transfer (drag force, wall lubrication force) and interfacial heat and mass transfer (condensation). Though the bubble size is a very important modeling parameter, in the present work a default bubble diameter has been used, which is modeled as a linear as a linear function of local liquid subcooling [2].

2. Alternatively the bubble-induced turbulence may be taken into account directly as additional terms in turbulent transport equations. In the present study these additional terms are implemented in $k$ and $\varepsilon$ equations:

$$\frac{\partial}{\partial t} \left( \alpha_i \rho_i k_i \right) + \nabla \cdot \left( \alpha_i \rho_i \bar{u}_i k_i \right) = \nabla \cdot \left( \alpha_i \frac{\mu_{\text{turb}}}{\sigma_k} \nabla k_i \right) + P_l - \rho_i \varepsilon_i + S_i^k,$$

$$\frac{\partial}{\partial t} \left( \alpha_i \rho_i \varepsilon_i \right) + \nabla \cdot \left( \alpha_i \rho_i \bar{u}_i \varepsilon_i \right) = \nabla \cdot \left( \alpha_i \frac{\mu_{\text{turb}}}{\sigma_{\varepsilon}} \nabla \varepsilon_i \right) + \frac{\varepsilon_i}{k_i} \left( C_{1\varepsilon} P_l - C_{2\varepsilon} \rho_i \varepsilon_i \right) + S_i^\varepsilon,$$

where $P_l$ is the production of turbulence due to the liquid shear stress. Two additional source terms corresponding to the bubble induced turbulence are:

$$S_i^k = -\vec{F}_D \cdot (\bar{u}_g - \bar{u}_l),$$

$$S_i^\varepsilon = C_{\varepsilon 3} \frac{S_i^k}{\tau},$$

where $\vec{F}_D$ is the interfacial drag force and $\tau$ is a characteristic time for bubble induced turbulence. The formulations by Troshko and Hassan [7]

$$\tau = \frac{2 C_{\text{vm}} d_b}{3 C_d |\bar{u}_g - \bar{u}_l|},$$

and by Yao and Morel [8]

$$\tau = \left( \frac{d_b^2}{\varepsilon_i} \right)^{\frac{1}{3}}$$

were used and compared. The coefficients in equation (7) are virtual mass coefficient $C_{\text{vm}}=0.5$ and drag coefficient according to Ishii and Zuber [9]. The coefficient $C_{\varepsilon 3}$ is 0.45 in case of Troshko and 0.45 or 0.1 in case of Yao formulation.

2.2 Interface momentum transfer

The interfacial transfer of momentum is modeled with the interfacial forces, which include drag force $\vec{F}_D$, virtual mass force $\vec{F}_{VM}$, lift force $\vec{F}_L$, turbulent dispersion force $\vec{F}_{TD}$ and wall lubrication force $\vec{W}_w$. The interfacial drag force is flow-regime dependent and is modeled according to a correlation by Ishii and Zuber [9]. To predict a non-homogeneous
radial void fraction distribution, the non-drag forces, which act perpendicularly to the flow direction, also need to be modeled. The lift force on the liquid phase can be calculated as

\[
\vec{F}_L = \alpha C_L \rho \left( \vec{u}_g - \vec{u}_l \right) \times \nabla \times \left( \vec{u}_l \right),
\]

where \( C_L \) is the lift force coefficient given by Tomiyama [10]. Lift force is shear-induced and pushes the bubbles towards the lower velocity region. The effect of dispersion of the vapor bubbles due the turbulent eddies in the liquid phase is taken into account by the turbulent dispersion force. In CFX-5 the turbulent dispersion force is based on the Favre averaging of the interfacial drag force [11]:

\[
\vec{F}_{TD} = -\frac{3C_D \mu_l}{4d_b \sigma_t} \left( \vec{u}_g - \vec{u}_l \right) \nabla \alpha, \tag{10}
\]

where \( C_D \) is the bubble drag coefficient, \( \mu_l \) total dynamic viscosity of liquid and \( \sigma_t \) is the turbulent Schmidt number for the liquid phase.

The contribution of the wall lubrication force in the subcooled boiling flow is probably the most difficult to evaluate, as it acts only on those near-wall bubbles which have already lift-off the wall. In the case, when there is some liquid flow between the bubble and the wall, the wall lubrication force acts in lateral direction away from the wall and prevents the accumulation of bubbles on the wall. The wall lubrication force approaches infinity as the wall distance approaches zero ensuring zero void fraction on the wall. However this cannot be applicable to the spherical cap bubbles growing and/or sliding on the heated wall during wall boiling. The wall lubrication force is commonly defined as:

\[
\vec{F}_W = -C_{Wall} \frac{\alpha \rho \left( \vec{u}_g - \vec{u}_l \right)^2}{d_b} \cdot \vec{n}, \tag{11}
\]

where \( C_{Wall} \) is wall force coefficient. Different models for \( C_{Wall} \) have been extensively tested for adiabatic bubbly flow however the conditions at bubbly flows with bubble generation at the wall are rather different. Two models for \( C_{Wall} \) were analyzed in the present study: a model by Antal et al. [12]

\[
C_{Wall} = \max \left( C_1 + C_2 \frac{d_b}{y_w}, 0 \right), \tag{12}
\]

with \( C_1 = -0.05 \) and \( C_2 = 0.01 \), and a model by Tomiyama [10]

\[
C_{Wall} = C_W \frac{d_b}{2} \left( \frac{1}{y_w^2} - \frac{1}{(D - y_w)^2} \right), \tag{13}
\]

with coefficient \( C_W \) depending on Eötvös number. Other variables \( y_w, d_b \) and \( D \) denote the distance from the wall, bubble diameter and pipe diameter, respectively.
2.3 Wall boiling model

During subcooled boiling flow, heat and mass exchange between the phases takes place on the heated wall and in the subcooled liquid flow. On the heated surface the vapour bubbles are generated and as they move through the subcooled liquid they condense and release the latent heat. The bubble condensation is briefly described in the following section. The evaporation mass flow on the wall $\dot{m}_w$ is applied to the near-wall cell and is modelled in a mechanistic way, taking into account the total mass of bubbles periodically departing from nucleation sites:

$$\dot{m}_w = \frac{\pi \cdot d_{bw}^3}{6} \rho_g f \cdot N_a,$$

where $d_{bw}$ is the bubble departure diameter, $f$ is the bubble departure frequency and $N_a$ is the nucleation site density. The boundary conditions for the heat transfer at the wall require a model for wall heat flux partitioning, which splits the total heat flux into the heat flux transferred to the liquid phase and the heat flux used to generate vapor. In the CFX-5 code a modified model of Kurul and Podowski [13] is implemented which splits the total wall heat flux into three different modes of heat transfer:

$$q_w = q_{\phi} + q_Q + q_e,$$

where $q_{\phi}$ is the single-phase convection heat flux transferred to the liquid phase near the wall outside the area influenced by nucleating bubbles $A_{hub}$, $q_Q$ is quenching heat flux transferred to the subcooled liquid from the bulk flow that fills the volume vacated by departing bubbles and $q_e$ is the fraction of the wall heat flux, that is directly used to generate vapor bubbles. The heat flux components are modeled as functions of local flow parameters, such as wall temperature $T_w$, liquid temperature $T_l$, latent heat $h_{lg}$, to just list a few. The single-phase convection heat flux is calculated using the single-phase blended linear-logarithmic temperature wall function of Kader [14]:

$$q_{\phi} = \frac{\rho_{cp} u_w}{T^+_{y^+(nw)}} \cdot (1 - A_{hub}) \cdot (T_w - T_{l(nw)}),$$

where $T_{l(nw)}$ is the liquid temperature of the near-wall computational cell, $T^+_{y^+(nw)}$ is analytically calculated non-dimensional temperature [14] at the non-dimensional distance from the near-wall cell $y^+(nw)$ and $u_w$ is the friction velocity. The single-phase convection heat flux takes place outside the area of nucleating bubbles $(1 - A_{hub})$. To obtain a grid independent solution the quenching heat flux is assumed to be proportional to the temperature difference at a given distance from the wall:

$$q_Q = h_Q A_{hub} (T_w - T_{l,y^+(const)}).$$

In Eq. (17), $h_Q$ is the quenching heat transfer coefficient and $T_{l,y^+(const)}$ is the liquid temperature at a given non-dimensional distance from the wall $y^+(const)$, which can be
predefined. Taking into account the self-similarity of non-dimensional temperature profiles at different $y^+$, the temperature difference in Eq. (17) can be calculated as:

$$(T_w - T_{l,y^+ (const)}) = \frac{T^+_{y^+ (const)}(T_w - T_{l,(nw)})}{T^+_{y^+ (nw)}}.$$  \hspace{1cm} (18)$$

The use of a temperature wall function with the boiling model is a novel approach implemented in CFX-5 [2] and will be discussed later. The evaporation heat flux can be derived from the evaporation mass flux

$$q_e = \dot{m}_w h_{lg}.$$  \hspace{1cm} (19)$$

The bubble influence area per unit wall area $A_{bub}$ is determined as

$$A_{bub} = \min \left[1, N_a K \left( \frac{\pi d_{bw}^2}{4} \right) \right]. \hspace{1cm} (20)$$

The parameter $K$ determines the size of the bubble influence area around the nucleation site on the heated wall that is subject to the quenching heat transfer. Commonly, the constant value of $K = 4$ is used. Thus, at high density of nucleation sites $N_a$ and large bubble size, the bubble influence area is formally limited by the total heating surface ($A_{bub} = 1$). The overlapping between the two neighboring sites with asynchronous nucleating bubbles is neglected. For bubble departure diameter the correlation of Tolubinski [15]

$$d_{bw} = \min \left[ 1.4 [mm], 0.6 [mm] \cdot \exp \left( -\frac{\Delta T_{sub}}{45[K]} \right) \right]$$

has been used. The upper limit for the bubble departure diameter ($d_{bw} = 1.4$ mm) is based on high-pressure experimental data. The variables $f$, $N_a$ and $h_Q$ are also calculated from empirical or semi-empirical equations which are basically the same as in the work of Kurul and Podowski [13]. The remaining unknown in equations for heat flux components (16, 17, 19) is the wall temperature $T_w$ that can be calculated from the wall heat flux balance (15) with an iterative procedure using a bisection algorithm.

The use of Kurul and Podowski partitioning model may be arguable, as it does not take into account the contribution to the evaporation heat flux due to bubble sliding on the wall. Namely, when the bubble slides along the wall it remains longer time in contact with the hot surface and continuously grows before it eventually lifts off the wall. Very recently, Kljenak et al. [16] and Basu and Dhir [17], proposed a heat flux partitioning model, which takes into account also the bubble sliding effect. However, this phenomenon is very important only at low pressures (1 to 5 bar), whereas at high pressure conditions applicable to power reactors it is of minor importance. The heat fluxes required for boiling incipience are much higher at high pressures therefore the bubble residence time during which the bubble may grow and slide is much shorter.
Near-wall treatment

The vapour phase at the wall boiling model is generated in the first near-wall cell. In the wall boiling model by Kurul and Podowski [13] the local variables (liquid velocity $u_l$, liquid temperature $T_l$) that appear in boiling correlations were taken from the near-wall cell. However, most of the constitutive relations (e.g. quenching heat flux, bubble departure diameter) are derived for one-dimensional thermal-hydraulic codes in terms of mean temperature and mean velocity. When these correlations are applied straightforwardly, as CFD boundary conditions simply by replacing averaged values by local ones, this would inevitably lead to the grid dependent solution. This works only for very coarse grids, where the first grid cell covers the entire boundary layer thickness. To ensure grid invariant solution in the CFX-5 code a characteristic temperature $T_{l, y^+ (\text{const})}$ was used instead of local near-wall temperature and was calculated from the analytical profile of the single-phase temperature wall function [14] at the given non-dimensional distance from the wall $y^+$. In the present study a constant value of $y^+ = 250$ is used. The proposed approach is a possible way to accommodate the existing correlations for CFD calculation. However, the use of single-phase wall function is questionable since it is known that the velocity and temperature profiles in the bubbly boundary layer deviate from single-phase counterparts [7],[18]. Further investigation and subsequent upgrade of the model used is therefore necessary.

2.4 Interfacial condensation

The interfacial condensation rate $\Gamma_{\text{cond}}$ across the phase boundary is defined as

$$\Gamma_{\text{cond}} = \frac{h_{lg} A_i (T_{\text{sat}} - T_l)}{h_{lg}}, \quad (22)$$

where $A_i$ is the interfacial area per unit volume and $h_{lg}$ is the interfacial heat transfer coefficient modeled by widely used Ranz-Marshall correlation [5].

3 RESULTS

To investigate the subcooled boiling in narrow channels of nuclear reactors, numerous experiments of high pressure boiling have been performed in the past. The most widely used experimental setups consist of an upward flow through uniformly heated tube. In the presented analysis several boiling experiments with upward water flow through the heated tube performed by Bartolomei [1] were simulated. In all calculated cases, the experimental tubes were 2 m long but had different inner diameters. The experiments were performed at different pressures, mass flow and heat flux conditions. The CFX-5 boiling model described in previous chapter was used for calculations. The effect of heat flux partitioning parameters, two-phase flow turbulence and non-drag forces were investigated in particular.

3.1 Validation against high-pressure experimental data

To obtain a grid independent solution a grid convergence analysis for three different uniform numerical grids was performed. The grid was refined in radial and circumferential directions. The $60^\circ$ sector of the tube was divided into 3 sub-blocks, in a way that the angle deformation for grid cells was minimised. The selected experiment was performed in a 15.4 mm vertical tube with the system pressure of 4.5 MPa, mass flow rate of 900 kg/ (sm$^2$) and heat flux of 0.57 MW/m$^2$. The comparison of averaged void fraction distributions in Figure 1
shows that the numerical grid 300x150 is sufficient to obtain a grid independent result. Here, the number 300 denotes number of grid cells over the cross-section plane of a 60° tube sector and the number 150 denotes number of grid cells in the axial direction.

Figure 1 Distributions radial void fraction and liquid temperature (Axial height = 1.2 m)

The calculated 3D distributions of void fraction and temperature for case 2 are presented In Figure 2 (a, b), whereas in Figure 2 (c), the calculated values of void fraction and temperatures are compared against the experimental data. A very good agreement between the calculations and measurements has been achieved for this experiment.

Figure 2 Calculated distributions of void fraction (a) and temperature (b) in the channel; (c) Measured vs. calculated averaged void fraction, averaged temperature, axis temperature and wall temperature

To investigate the range of validity of the subcooled boiling model at different pressure, mass flow and heat flux conditions, numerous experiments performed in a 12 mm tube were calculated. The averaged void fraction with respect to thermodynamic quality is presented in Figures 3 to 6. The pressure effect is presented in Figure 3. A reasonably good agreement
with experiments was obtained at pressures between 3 to 11 MPa, whereas the void fraction is underpredicted at 15 MPa. The under-estimation of void fraction was found at mass fluxes higher than 1000 kg/(m²s) as may be observed in Figure 4. The effect of heat flux for different inlet mass flux and pressure conditions is shown in Figure 5 and 6. For lower mass flux \((G=1000 \text{ kg/(m}^2\text{s})\) a good agreement with experiment can be observed at the low heat fluxes, whereas the code underpredicts the data at higher heat fluxes. In the case of higher mass flux \((G=2000 \text{ kg/(m}^2\text{s})\) the void fraction is over-predicted at lower heat fluxes and under-predicted at higher heat fluxes. Large deviation from measured data at high mass flow rates \((2000 \text{ kg/m}^2\text{s})\) and at high heat fluxes \((2 \text{ MW/m}^2\)\) may be attributed to the fact, that not all the phenomena (bubble sliding and merging on the wall, bubble coalescence in the bulk flow) are included in the wall boiling model.

3.2 Turbulence modelling

During analysis it was observed that the choice of the turbulence model has a strong effect on the void fraction distribution. Therefore, a sensitivity study was performed, where the k-\(\omega\) based SST model and the standard k-\(\varepsilon\) model for the liquid phase turbulence were compared. The Bartolomei experiment carried out in a tube with the diameter of 24 mm was simulated, since at this experiment, the radial measurements of the temperature were also available. In both cases Sato model for bubble induced turbulent viscosity was applied. To isolate the effect of turbulence model, the lift and wall lubrication forces were not used. The turbulent dispersion force was still included, in order to assure a converged solution.
The calculation with SST turbulence model shows better agreement against averaged void fraction, whereas the k-ε model significantly over-predicts the measured data, especially at low void fraction values.

Modelling of bubble induced turbulence

The Sato approach (Eqs. 1 and 2) with an additional viscosity term is widely used to describe bubble-induced turbulence. However, it was found that this formulation results in very low values of turbulent kinetic energy $k$, even lower than in the case without any bubble-induced model (denoted as “no Sato”). The correct prediction of $k$ value is very important especially at bubble coalescence and break-up, as the value of turbulent kinetic energy appears as a parameter in these models. Though they should be considered at high void fractions, the bubble coalescence and break-up models are not yet tried in the present subcooled boiling model. The Sato approach is compared against the calculation without any bubble-induced model and against models of Troshko [7] and Yao [8] in Figures 10 and 11, where the bubble-induced turbulence in liquid is taken into account via additional source terms in $k$ and $ε$ equations. Models of Troshko and Yao are described in previous section by Eqs. (7) and (8), respectively. In addition, in case of Yao model, two different values of coefficient $C_{3ε}$ are used. To isolate the effect of different formulations the non-drag forces (except turbulent dispersion) were not activated in these calculations.

Figure 7 Influence of liquid phase turbulence model

Figure 8 Influence of bubble-induced turbulence model: turbulent kinetic energy and overall eddy viscosity
Figure 9 Influence of bubble-induced turbulence model: liquid temperature and void fraction

In general, models of Troshko and Yao predict significantly higher $k$ values than the Sato model. Considering the variation of $C_\varepsilon$ coefficient in Yao formulation, it can be observed that the lower value of $C_\varepsilon$ produces significantly higher overall eddy viscosity and adds somewhat more diffusion towards the pipe centre (see temperature distributions). Just near the wall, the Sato model predicts the highest temperature diffusion due to bubble induced turbulence, while the other models give somewhat more diffusion towards the pipe center.

3.3 Influence of non-drag forces

Separate effects of non-drag forces in boiling (turbulent dispersion force, lift force, wall lubrication force) were investigated. In all cases, the SST turbulence model with Sato bubble-induced turbulence was used. As may be observed in Figures 12 to 13, the wall lubrication force has the largest effect on the void fraction distribution.

Figure 10 Influence of Non-drag forces; a.) Axial averaged void fraction b.) Radial profiles of liquid temperature along the channel
a.) Figure 11 Influence of non-drag forces; a.) Radial void fraction profile at 1.45 m b.) Radial liquid velocity profile at 1.45 m

To estimate the effect of different wall force models, two different correlations (Tomiyama and Antal) for wall lubrication force were implemented and compared. In both cases, the turbulent dispersion (FAD) model and the lift force (Tomiyama) model were also used. For the liquid phase, the SST turbulence model with Sato bubble-induced turbulence was used.

b.) Figure 12 Different wall force models: Wall force value distribution and void fraction profile

The use of wall lubrication force is questionable for the bubbles still growing on the wall since according to its formulation it requires some amount of liquid between the bubble and the wall. In the case of wall boiling, Tomiyama formulation (proportional to 1/y^2) of wall lubrication force results in an extremely high value, where values of other non-drag forces are practically negligible. Better choice is to use Antal formulation with reciprocal linear relation to the wall distance. Both the linear (1/y) and the quadratic models (1/y^2) formally contradict with the wall source of bubbles, since their integrals across the boundary layer are infinitely large with the non-zero volume fraction at the wall.

4 CONCLUSIONS

A wall boiling model implemented in the state-of-the-art two-fluid model of CFX-5 code was used here. Grid independent solution on dense grids was obtained by using adequate near-wall treatment, based on the analytical temperature wall function. The simulation results have been validated against published experimental data of subcooled flow boiling in a...
vertical pipe relevant to pressurized water reactor conditions. In general, the calculated averaged void fraction distributions show reasonable agreement with most of the experimental data. It may be concluded that the presented subcooled boiling model is best suited for pressure range (3 to 11 MPa), mass flow rate of about 1000 kg/m$^3$s and for heat fluxes up to 1.2 MW/m$^2$.

To investigate the effect on lateral distributions of void fraction and liquid temperature the modelling of turbulence and non-drag forces under wall boiling conditions was also studied. It has been shown that it is necessary to model the effects of bubble induced turbulence and non-drag forces for realistic simulation of the two-phase flow field. However, this is still a subject of further modelling research, which will be focused towards derivation of an adequate formulation of wall force and bubble induced turbulence.

ACKNOWLEDGMENTS

The financial support from the Slovenian Research Agency for the project “Three-dimensional Eulerian model of convective boiling” (Z2-6651) is gratefully acknowledged.

REFERENCES


