WALL-TO-FLUID TRANSFER MECHANISMS IN BOILING FLOWS

Boštjan Končar, Mavko Borut

Jožef Stefan Institute,
Jamova 39, SI- 1000 Ljubljana, Slovenia
bostjan.koncar@ijs.si**, **borut.mavko@ijs.si

ABSTRACT

To better understand and predict the boiling flow processes, accurate two-fluid numerical models are needed. One of the important goals of the NURESIM project is to assess and improve the simulation capability of the three-dimensional two-fluid codes for prediction of local boiling flow processes. The boiling flow is strongly affected by local mechanisms in the turbulent boundary layer near the heated wall. Since the computational grid of the 3D two-fluid models is too coarse to resolve the variable gradients in the near-wall region, the use of wall functions is a common approach to model the liquid velocity and temperature profile in adjacent to the heated wall.

The main objective of this work was to develop the wall function model for the boiling boundary layer. The wall function model for momentum, based on the surface roughness analogy has been developed and implemented in the V1.0.6 version of the NEPTUNE_CFD code. The model has been validated on several upward boiling flow experiments, differing in the geometry and working fluid. The simulations with the new wall function model show an improved prediction of flow parameters in the boiling boundary layer. Furthermore, a generic wall function for the energy equation has been derived but at the present phase it was not yet implemented in the NEPTUNE_CFD code.

1. INTRODUCTION

The heat transfer and phase-change mechanisms in the subcooled flow boiling are governed mainly by local mechanisms in the boundary layer adjacent to the heated wall (bubble nucleation, bubble growth, sliding and detachment of bubbles, bubbles merging on the wall, rewetting after detachement, etc.).

For computational simulations of realistic bubbly flows, the use of Eulerian two-fluid models is still the most appropriate. Traditionally, two-fluid models employ eddy viscosity closure models (e.g. k-ε) to describe Reynolds stress in the liquid phase and rely on the single-phase logarithmic law of the wall as a near-wall boundary condition. The grid of the multidimensional two-fluid models is too coarse to resolve the variable gradients in the boundary layer therefore the use of wall functions is a common approach to describe the liquid velocity and temperature in the near-wall region. In the case of boiling boundary layer with high concentration of bubbles generated on the heated wall, the use of single-phase wall law fails to predict the velocity profile in the near-wall region. In the literature there are only a few boundary layer models for boiling flow, but these are more or less empirical wall functions, obtained by fitting the coefficients in the log-law equation to the experimental data (e.g. Roy and Zarate (2002)). Therefore a generic wall function, based on thermo-physical mechanisms of nucleate boiling is presented in this work. A wall function model based on the surface roughness analogy is proposed for the momentum equation. The increase of shear velocity due to bubble "surface roughness" also leads to the increased heat transfer coefficient in the single-phase heat flux component of the heat flux partitioning model. Here, only a derivation of an enhanced heat transfer coefficient influenced by enhanced two-phase wall friction is presented. The new wall function for the momentum equation has been implemented in the V1.0.6 version of the NEPTUNE_CFD code.

2. PHYSICAL MODEL IN THE NEPTUNE_CFD CODE

The basic model of NEPTUNE_CFD is the classical six-equation two-fluid model together with k-ε transport equations used for modelling of the liquid phase turbulence (Lavieville et al., 2005). In boiling flow the

liquid phase is dominant and is treated as a continuous, whereas the vapor bubbles are modeled as a dispersed phase. The effect of bubble wakes on the liquid turbulence is taken into account by additional terms in *k-ε* transport equations. The interfacial transfer of momentum is modelled by interfacial forces per unit volume, which include drag force, added mass force and the non-drag forces (lift, turbulent dispersion force), which act perpendicularly to the flow direction. The interfacial heat and mass transfer due to condensation in the subcooled bulk flow was modeled by Ranz-Marshall correlation (Ranz and Marshall, 1952). Detailed description of mass and heat transfer models is given in the code theory manual (Lavieville et al., 2005). Here, only wall-to-fluid transfer models, relevant for computations of boiling flows are described. The version V.1.0.6 of the NEPTUNE_CFD code has been used in this work.

2.1. Wall-to-fluid heat transfer at subcooled boiling

To model wall-to-fluid heat transfer at subcooled flow boiling a two-step approach is used in the NEPTUNE_CFD code. The two steps include calculation of the condition for boiling incipience in terms of critical wall superheat (see Lavieville et al., 2005) and calculation of heat flux partitioning. To describe wall-to-fluid heat transfer an approach of Kurul and Podowski (1990) is adopted, which splits the total wall heat flux into three components:

$$\Phi_w = \Phi_{C1} + \Phi_Q + \Phi_E , \tag{1}$$

where Φ_{C1} denotes the single-phase convection heat flux to the liquid, Φ_Q denotes quenching heat flux that transfers cold liquid from the bulk flow to the wall periodically and Φ_E is the heat flux component needed to generate vapor bubbles. Heat flux components are calculated as follows:

$$\Phi_{C1} = A_C h_{\log} \left(T_w - T_\delta \right), \tag{2}$$

$$\Phi_Q = A_Q t_Q f \frac{2\lambda_1 \left(T_w - T_\delta \right)}{\sqrt{\pi a_l t_Q}}, \tag{3}$$

$$\Phi_E = \frac{\pi d_{bw}^3}{6} f \rho_g N_a h_{\lg} . \tag{4}$$

The wall surface area per unit volume is split into two parts: an area influenced by nucleating bubbles A_Q and a "single-phase convection" area A_C unaffected by the bubbles. They are related by expression $A_Q + A_C = 1$. Parameters in equations (2) to (4) are the following: h_{log} is heat transfer coefficient in thermal boundary layer, τ_Q is the quenching period between the bubble departure and beginning of the growth of a subsequent one, f is bubble departure frequency, λ_1 is liquid thermal conductivity, a_l is the liquid thermal diffusivity, d_{bw} is maximum bubble diameter at departure, N_a is density of active nucleation sites and h_{lg} is latent heat for evaporation. The correlations used to calculate these parameters are described in the code manual (Lavieville et al., 2005) and in our previous references (Koncar, 2007b). Bubble departure diameter is calculated by the extended Unal model.

In Eqs. (2) and (3), T_δ denotes a characteristic liquid temperature in the turbulent boundary layer. In the previous version of the NEPTUNE_CFD the liquid temperature in the centre of the near-wall cell was adopted. To obtain the grid independent solution, the temperature T_δ is calculated from the analytical profile of the two-phase temperature wall function at the given non-dimensional distance from the wall y^+_δ. Taking into account the self-similarity of non-dimensional temperature profiles at different y$^+$, the temperature T_δ reads:

$$T_\delta = T_w - \frac{T^+(y^+_\delta)}{T^+(y^+_{FC})} \left(T_w - T_{l,FC} \right), \tag{5}$$

where subscript *FC* denotes the first near-wall cell and δ given distance from the wall.

Similar approach is adopted also for characteristic velocity u_δ, which appears in Unal correlation (Unal, 1976) for bubble departure diameter d_{bw}:

$$u_\delta = \frac{u^+(y^+_\delta)}{u^+(y^+_{FC})} u_{l,FC}, \tag{6}$$

To define a fixed location in the turbulent boundary layer, a constant value of $y^+=250$ was used in the present study. This model has already been included in the CFX code (Wintterle et al., 2005; Koncar et al., 2007a) and recently also in the NEPTUNE_CFD code (Vyskocil and Macek, 2007). It should be noted that this approach is valid only if near-wall cells remain in log region of the boundary layer ($30 < y^+ \leq 300$).

3. WALL FUNCTIONS FOR BOILING FLOW

Wall functions use empirical laws to overcome the inability of the k–ε model to predict a logarithmic velocity profile near a wall. With these laws it is possible to express the mean velocity parallel to the wall and turbulence quantities outside the viscous sublayer in terms of the distance to the wall and wall conditions such as wall shear stress, pressure gradient and wall heat transfer. In this study only the wall functions used to provide near-wall boundary conditions for the momentum and turbulence transport equations are considered. The state-of-the-art CFD codes rely on the standard single-phase wall functions to model the two-phase boundary layer. At subcooled flow boiling, the liquid velocity profile in the boiling boundary layer is significantly disturbed by the bubble formation and detachment mechanisms on the heated wall. In our previous work (Končar et al., 2005a, Končar et al., 2005b) and in the work of other researchers (Lee et al., 2002; Morel et al., 2005) an over-prediction of liquid and gas velocity distributions in the boiling boundary region has been reported. The use of single-phase log-law was recognized to be one of the main reasons for discrepancy.

In our previous work (Končar, 2007b), two different wall function models have been implemented (in CFX-4) and compared: one for non-boiling bubbly flow (Troshko and Hassan, 2001) and the other for boiling flow, which treats the bubbles as an artificial surface roughness (Ramstorfer et al., 2005). The approach of Troshko and Hassan (2001) depends on void fraction as a key input parameter, but it cannot be easily extended to the case of subcooled boiling flow. It was shown that the surface roughness analogy is a good solution to mimic the effect of nucleating bubbles on the liquid turbulence in the boundary layer.

3.1. Model of artificial surface roughness due to bubble nucleation

Surface roughness analogy for modelling of boiling boundary layer has been first used by Ramstorfer et al. (2005). The main idea of the new wall function is that nucleating bubbles on the wall disturb the boundary layer flow in a similar way as the surface roughness. As a basis, a logarithmic law for turbulent flows over rough walls is used

$$u^+ = \frac{1}{\kappa}\ln(y^+) + B - \Delta u^+. \tag{7}$$

where velocity $u^+ = u_t / u_w$ and distance from the wall $y^+ = \rho_l u_w \Delta y / \mu_l$ are written in non-dimensional wall units scaled by wall friction velocity $u_w = \sqrt{\tau_w / \rho_l}$ (τ_w is the wall shear stress). Here u_t is the known velocity tangential to the wall and Δy is the distance from the wall. Coefficients κ and B are standard single-phase constants with the values of 0.41 and 5.3, respectively. The last term in Eq. (7) represents the offset of u^+ due to the wall roughness:

$$\Delta u^+ = \begin{cases} \frac{1}{\kappa}\ln(1 + C_{kr} k_r^+); & k_r^+ > 11.3 \\ 0; & k_r^+ \le 11.3 \end{cases}, \tag{8}$$

where C_{kr} is a roughness constant, which depends on the type of roughness (C_r=0.5 for sand-grain roughness) and k_r^+ is the roughness Reynolds number:

$$k_r^+ = \frac{\rho_l k_r u_\tau}{\mu_l}. \tag{9}$$

The quantity k_r represents the physical roughness height of the surface. For $k_r^+ \le 11.3$, the wall is considered to be smooth, otherwise the wall is treated as rough. Although Ramstorfer and co-workers (2005) have studied the flow boiling in a horizontal channel this type of log-law may be applied to all boiling flows where the flow motion along the wall is dominant. The model assumes that for boiling on the heated surface the roughness height can be represented by a functional dependence on the bubble departure diameter d_{bw} and the contribution of nucleate boiling heat flux Φ_{nb} to the total heat flux Φ_w:

$$k_r = \eta d_{bw}\left(\frac{\Phi_{nb}}{\Phi_w}\right)^\varsigma = \eta d_{bw}\left(1 - \frac{\Phi_C}{\Phi_w}\right)^\varsigma. \tag{10}$$

In this study, bubble departure diameter is calculated according to the extended Unal model (Lavieville et al., 2005). The ratio of the nucleate boiling component to the total heat flux Φ_{nb}/Φ_w takes into account the thickening of the boiling boundary layer with increasing boiling activity. The coefficients η and ς in Eq. (10) are empirical parameters set to the values η=0.5 and ς=0.174 for the purpose of this study. The proposed "boiling" law of the wall is implemented in the NEPTUNE_CFD code in the form of blended linear-logarithmic wall function as follows:

$$u^+ = \min\left(y^+, \frac{1}{\kappa}\ln(y^+) + B - \Delta u^+\right). \tag{11}$$

When implementing the wall function model into the computer code, somewhat different scaling parameters need to be defined. For example, Eq. (8) becomes singular if u_t approaches zero. That may occur at separation points in boundary layer. Therefore in the logarithmic region, an alternative scaling parameter, u*, which is defined by the means of turbulent kinetic energy k_l can be used instead:

$$u^* = c_\mu^{1/4} k_l^{1/2}, \tag{12}$$

This parameter doesn't go to zero with u_t, as in turbulent flows k_l is never completely zero. The scaled distance from the wall is then

$$y^* = \rho_l u^* \Delta y / \mu_l. \tag{13}$$

Based on this definition the wall shear velocity u_w and the wall shear stress τ_w can be calculated as

$$u_w = \frac{u_t}{\frac{1}{\kappa}\ln(y^*) + B - \Delta u^+}, \tag{14}$$

$$\tau_w = \rho_l u^* u_w. \tag{15}$$

From shear velocity, near-wall values for momentum equations can be calculated. The new wall function was implemented into the code NEPTUNE_CFD V.1.0.6.

3.2. Influence of two-phase wall friction on the heat transfer in wall boiling model

The wall-to-fluid heat transfer in boiling flows may be influenced by enhanced two-phase wall friction. The shear velocity may influence the wall-to-fluid heat transfer in terms of enhanced heat transfer coefficient of the convective component Φ_{C1} of the wall heat flux. To describe the enhancement of the convective heat flux component in the boiling flow, the Colburn analogy between the wall heat transfer and the wall shear velocity can be used:

$$\Phi_{C1,2ph} = \Phi_{C1,1ph} \cdot \left(\frac{u_{w,2ph}}{u_{w,1ph}} \right)^2 , \tag{16}$$

where $\Phi_{C1,2ph}$ is the convective heat flux component for boiling flow, whereas $\Phi_{C1,1ph}$ is the convective heat flux in the single-phase flow. If available, the ratio ($u_{w,2ph}$ / $u_{w,1ph}$) may be determined from the experimental data by relating the measured wall shear velocities for the boiling two-phase flow to those measured in the corresponding bubble-free single-phase flow. Otherwise, the ratio ($u_{w,2ph}$ / $u_{w,1ph}$) can be calculated by relating the corresponding wall functions:

$$\left(\frac{u_{w,2ph}}{u_{w,1ph}} \right)^2 = \left(\frac{u_t \Big/ \frac{1}{\kappa} \ln(y^+) + B - \Delta u^+}{u_t \Big/ \frac{1}{\kappa} \ln(y^+) + B} \right)^2 = \left(\frac{\frac{1}{\kappa} \ln(y^+) + B}{\frac{1}{\kappa} \ln(y^+) + B - \Delta u^+} \right)^2 . \tag{17}$$

The two-phase wall function for boiling flow (Eq. 14) is in the denominator of Eq. (24), whereas the expression in the numerator represents the single-phase log-law. The Eq. (9) for convective heat flux can be rewritten in terms of enhanced convective heat transfer due to the increased two-phase wall shear velocity:

$$\Phi_{C1,2ph} = A_C h_{\log} \left(\frac{u_{w,2ph}}{u_{w,1ph}} \right)^2 \left(T_w - T_\delta \right). \tag{18}$$

The heat transfer coefficient for single-phase convection h_{log} is given by:

$$h_{\log} = \frac{\rho_l c_{pl} u_{w,1ph}}{T^+_{\delta,1ph}}, \tag{19}$$

where $u_{w,1ph}$ is the single-phase shear velocity and $T^+_{\delta,1ph}$ is a non-dimensional liquid temperature in the single-phase thermal boundary layer. The analytical expression for the single-phase temperature wall function $T^+_{\delta,1ph}$ is given in the paper of Morel (2005) and is already used in the NEPTUNE_CFD code. Here, merely a derivation of the enhanced convective heat flux is presented, but it has not yet been implemented into the NEPTUNE_CFD code.

4. RESULTS

Three sets of experimental data from the literature were used to validate the boiling model of the NEPTUNE_CFD code. Arizona State University (ASU) experiments and Korean Atomic Energy Institute (KAERI) experiments were both performed in an annular vertical channel, whereas DEBORA experiments (performed at CEA) were performed in a vertical pipe. Furthermore, different working fluids have been used in different experimental facilities. In ASU experimental facility the refrigerant R-113 is used, DEBORA facility

uses R-12, while water at the low pressure (up to 2 bar) is used in KAERI test section. Density ratios and other scaling numbers are therefore different.

4.1. Experimental data

The measurement section of the ASU experimental facility (Roy et al., 2002) consists of a vertical annular channel with a heated inner tube outer diameter of 15.8 mm and insulated outer tube with inner diameter of 38.02 mm. The total length of the annulus is 3.66 m and the 2.75 m long upper part of the inner tube is heated by the direct current. The 0.91 m long lower part of the annulus is not heated. The local measurements of transversal profiles of void fraction, phase velocities, velocity fluctuations and liquid temperature were performed at a single axial location 1.99 m downstream from the beginning of the heated section. The measurement probes and measurement techniques used in ASU experiments are described in the original paper of Roy et al. (2002).

The second set of experiments in KAERI facility was performed by Lee et al. (2002). The vertical annular test channel in KAERI facility is 2.376 m long with a heated inner tube. The 1.67 m long inner tube with outer diameter of 19 mm is composed of a heated section with copper electrodes at both ends of the heated section. The outer tube consists of two stainless steel tubes of 37.5 mm inner diameter, which are connected below the measuring plane by a transparent 50 mm long glass tube. Local measurements of void fraction, phase velocities and bubble size over the channel cross-section were performed 1.61 m downstream from the beginning of the heated section. The temperature and absolute pressure were measured at the inlet.

The DEBORA experiments were carried out in vertical pipe test section at CEA (Morel, 2003; Garnier, 2006). The refrigerant R-12 has been adopted as the working fluid to simulate the Pressurized Water Reactor (PWR) conditions under low pressure. The fluid flows upwards inside a vertical pipe having an internal diameter equal to 19.2 mm. The whole pipe can be divided axially into three parts: the adiabatic inlet section (1m length), the heated section (3.5m length) and the adiabatic outlet section (≈0.5m length). At the end of the heated section, the radial profiles of void fraction, gas velocity and bubble diameter have been measured by means of an optical probe. ASU, KAERI and DEBORA experimental conditions are presented in Table 1.

Table 1 ASU (Roy, 2002), KAERI (Lee, 2002) and DEBORA (Morel, 2003; Garnier, 2006) experimental conditions

Exp. Facility	Exp. No.	Working fluid	$p_{m.p.}$ (bar)	q_w (kW/m^2)	G (kg/m^{2}s)	T_{sat} (^{o}C)	T_{inlet} (^{o}C)
ASU	tp1	R-113	2.69	95	568	80.5	42.7
	tp6	R-113	2.69	116	784	80.5	50.2
KAERI	Lee 1	Water	1.15	169.76	478.14	103.5	83.9
	Lee 3	Water	1.30	114.78	476.96	107.1	95.6
DEBORA	DEB1	R-12	26.15	73.89	2064	86.65	68.52
	DEB2	R-12	30.06	58.26	1007.4	94.14	58.39

4.2. Grid refinement analysis on ASU experiment

The ASU experiment *tp6* was selected to evaluate the grid sensitivity of the NEPTUNE_CFD simulations with the new velocity wall function. Three different 2D equidistant grids were tested (10x110, 20x220, 30x220), where the lower number denotes the number of cells in radial direction and the higher number denotes the number of cells in the stream-wise direction. The y^+ values of the near-wall points at the measuring location (2.9 m) lie in the turbulent boundary layer for all grids. Simulations of void fraction, liquid velocity and liquid temperature for three different grids are presented in Figure 1. On the coarse grid, NEPTUNE_CFD predicts significantly different values of all variables in the near-wall region than for both finer grids. Discrepancy

between both finer grids can be considered as negligible, therefore the "medium size" grid 20 by 220 was used for further computations.

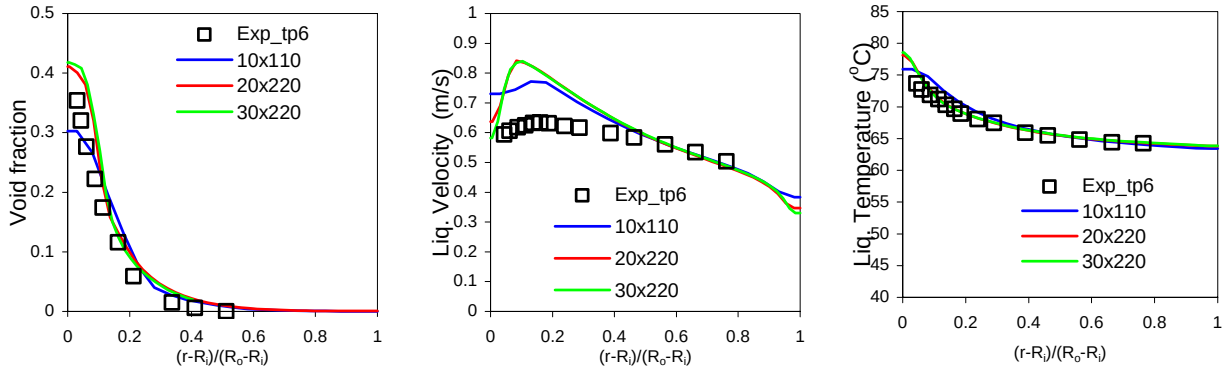

Figure 1: Grid sensitivity – with new wall function model

4.3. Validation against experiments

In Figures 2 to 6 the effect of boundary layer model is analysed. Experimental cases from Table I were simulated. The calculations were performed with the version 1.0.6 of the NEPTUNE_CFD code. The calculations differ in wall function models. The "base" calculation uses standard single-phase law of the wall, whereas "Wall func" calculation uses modified wall function model as described by Eq. (14). Other models, described in section 2 are the same for both calculations.

Radial void fraction profiles are shown in Figure 2. For the two ASU cases, the "wall func" simulation predicts somewhat higher void fraction and broader two-phase region comparing to the base calculation, but the agreement with experimental data is very good. In the case of KAERI experiments, the predicted two-phase regions are much narrower than experimental ones and the void fraction peak is higher. The turbulent dispersion force at KAERI experiments is too low, which results in a too weak diffusion of bubbles towards the centre of the channel and their accumulation on the wall. A constant value of turbulent dispersion coefficient C_{TD}= 1 seems to be too low for water boiling flow at KAERI experiments. The agreement of void fraction profiles is acceptable for DEBORA experiments – the void fraction profile is more gradual in the case of modified wall function model.

The calculation of stream-wise gas velocity depends on the model for interfacial drag and interfacial area density (e.g. bubble size) whereas the stream-wise liquid velocity profile in the wall boundary layer mainly depends on the wall friction, determined by the velocity wall function. Other influencing parameters are non-drag forces. The liquid and gas stream-wise velocities are compared in Figures 3 and 4. The "base" calculation significantly over-predicts measured liquid and gas velocities for both ASU cases. In the case of KAERI experiments, "base" liquid velocities are slightly under-predicted, but "base" gas velocities show high over-prediction. This may be to a large extent attributed to the over-prediction of void fraction. For DEBORA experiments only gas velocities were measured. In general, the calculation with the new wall function model improved the correspondence with the measured velocities near the wall. In the case of annular geometry (ASU and KAERI facilities), liquid velocity adjacent to the wall is decreased and gets closer to the measured data, whereas the over-prediction somewhat away from the wall is still notable. Due to the coupling through the interfacial drag a similar trend of decreased velocity near the heated wall may be observed also for the gas phase (Figure 4). For the pipe flow (DEBORA), the gas velocity profile shifts from wall-peaked to central-peaked which is in accordance with the measured gas velocity profile.

Turbulent kinetic energy profiles are presented in Figure 5. Measured values are available only for ASU experiments. ASU experiments show that turbulent kinetic energy is the highest in the boiling region near the inner wall and then rapidly decreases towards centre of the channel. This trend was not adequately reproduced by the "base" calculation, where k_l profile tends to be more gradual. As may be observed also for other simulated cases, the "wall func" calculation produced more turbulent kinetic energy in the near-wall region than the "base"

calculation. The liquid temperature profiles are compared in Figure 6. Liquid temperatures were measured for both ASU experiments and for the first DEBORA experiment. A good agreement for ASU experiments has been achieved for both calculations. The temperature profile for DEB1 case is very close to saturation temperature. Also here, a reasonable agreement between both calculations and experiment (± 2 ^{o}C) is demonstrated.

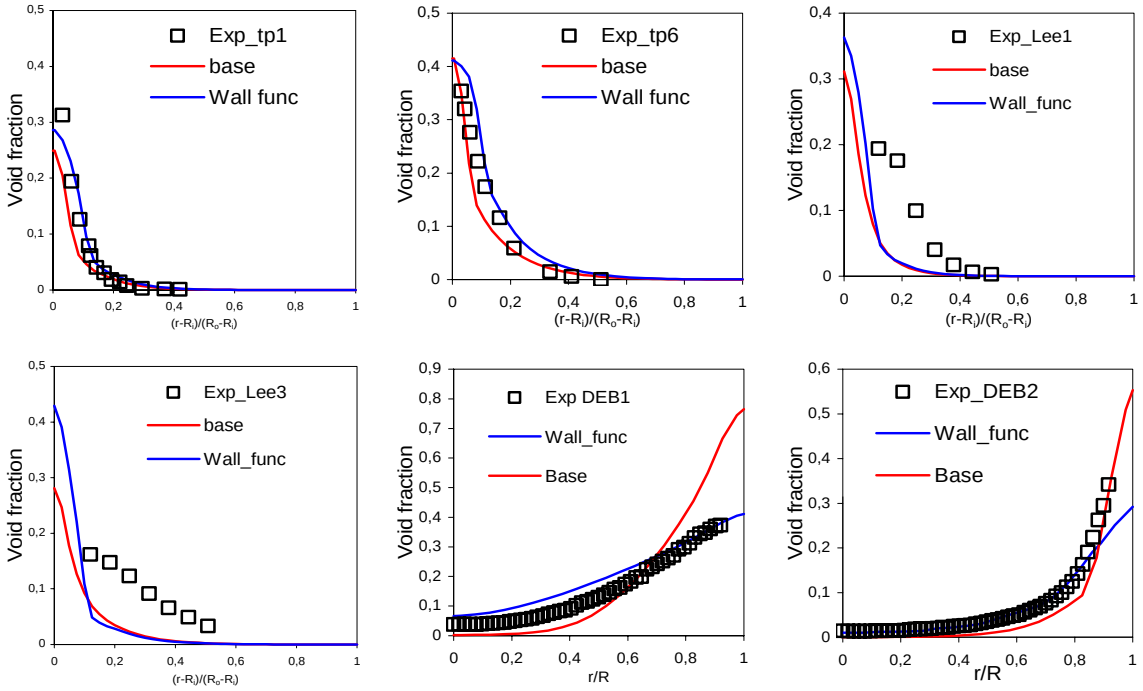

Figure 2: Void fraction profile

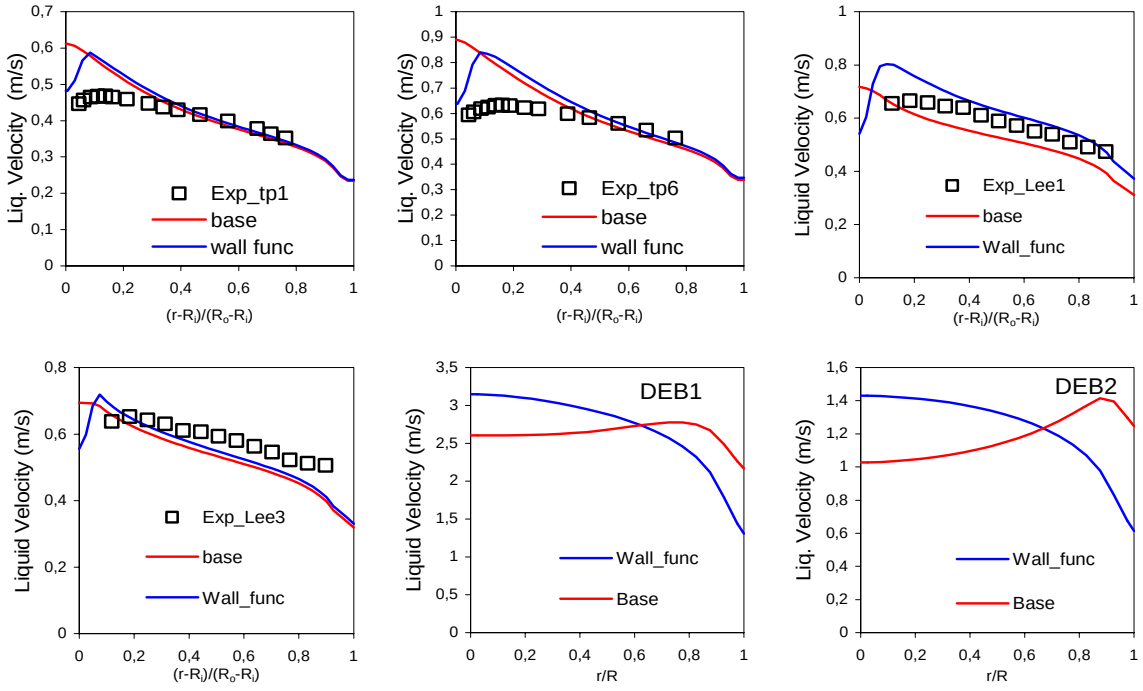

Figure 3: Liquid velocity profile

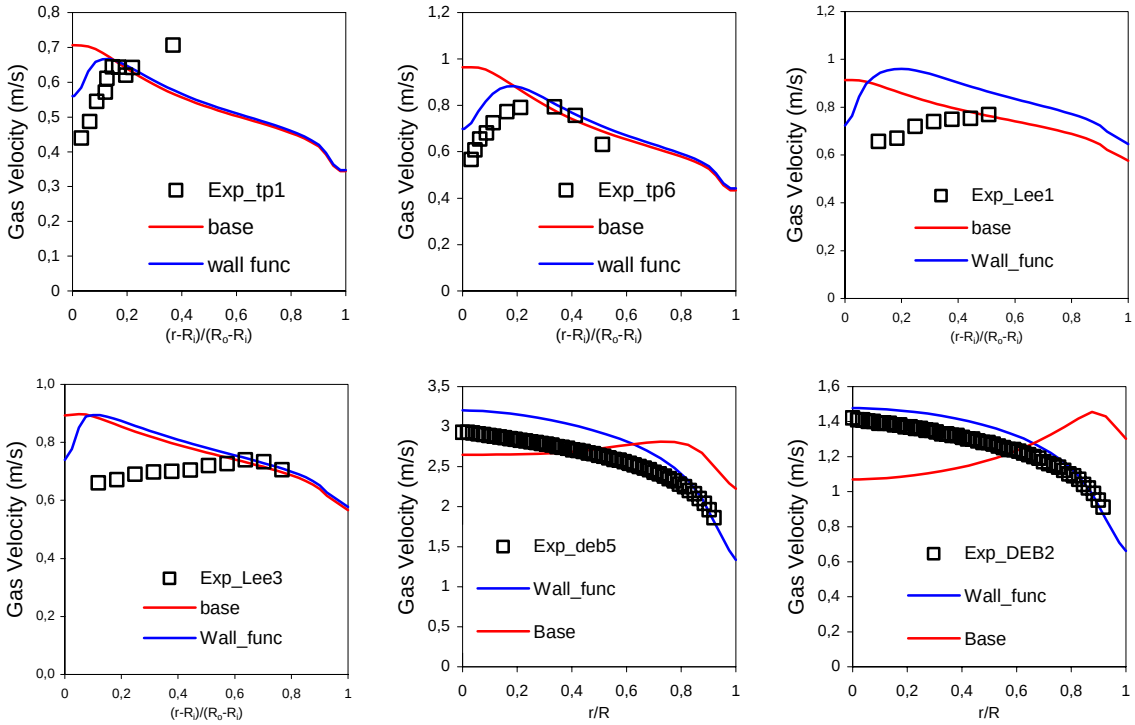

Figure 4: Gas velocity profile

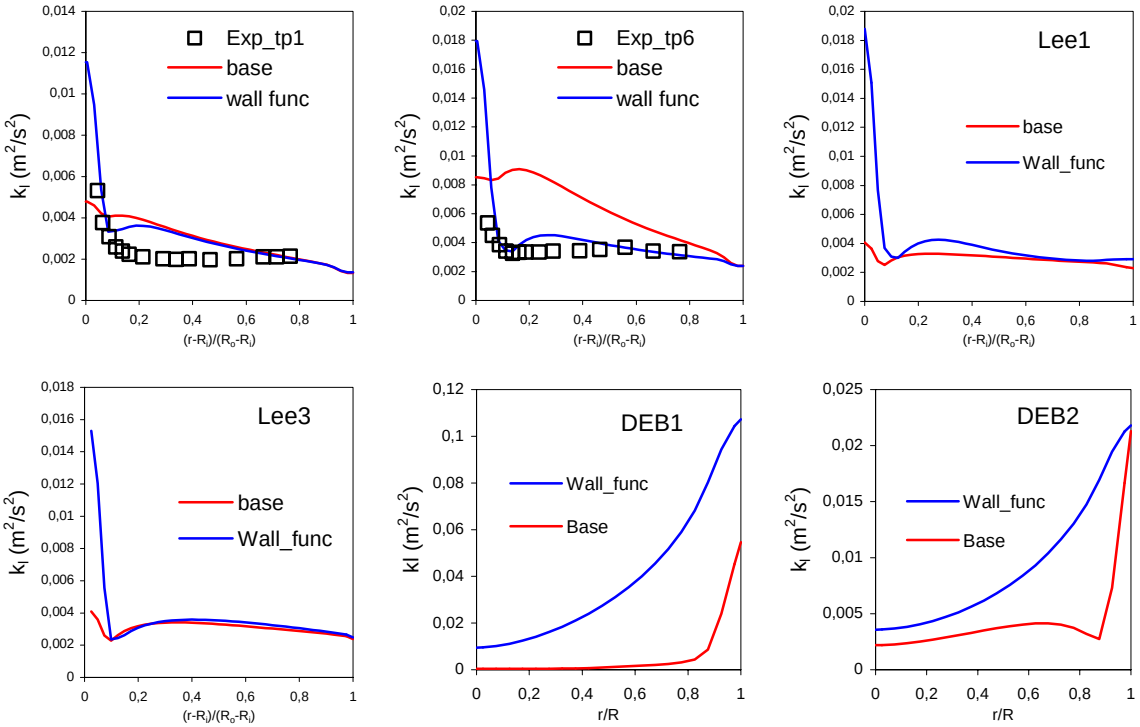

Figure 5: Turbulent kinetic energy profile

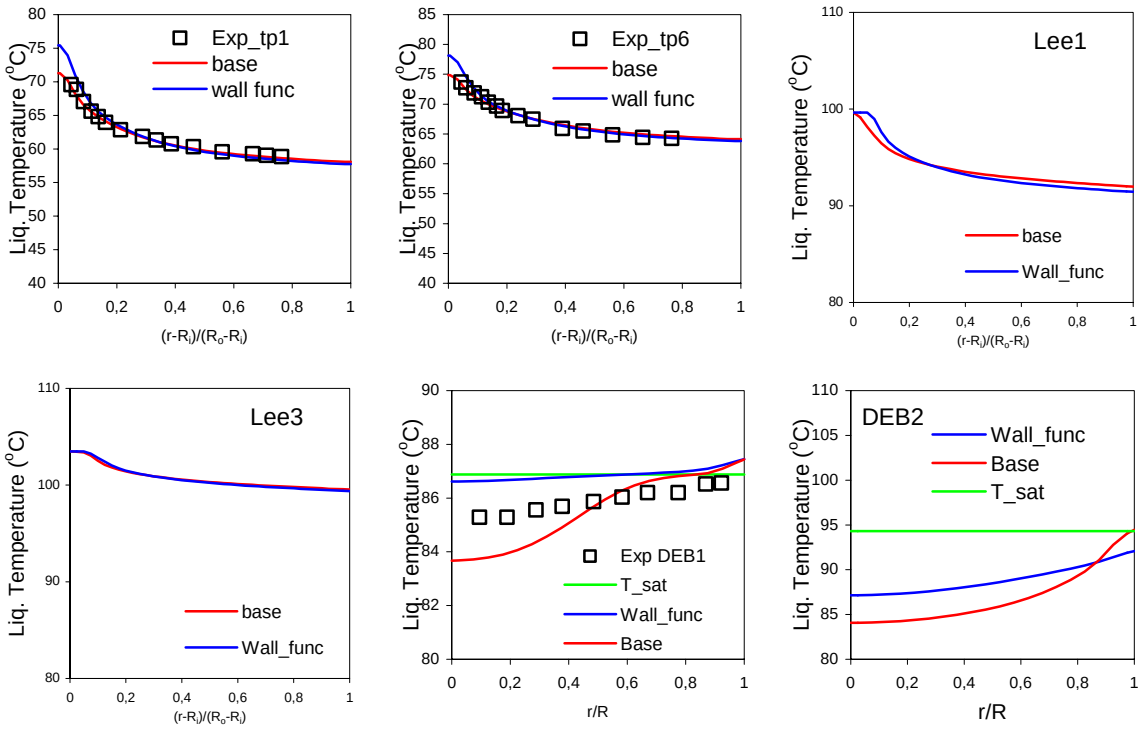

Figure 6: Liquid temperature

To check the validity of wall boiling model, the distribution of wall boiling parameters is presented in Figure 7. Figure 7a shows the evolution of the calculated heat flux components along the heated channel for the ASU *tp6* experiment. Although the subcooling is relatively high (~30°C), the applied heat flux is high enough to initiate boiling already at the beginning of the heated section. The single-phase convection heat flux decreases rapidly, while quenching and evaporation heat fluxes increase. When convective heat flux disappears a partially developed boiling changes to fully developed boiling, where the heated surface is completely covered by nucleating bubbles. In the fully developed boiling regime the evaporation heat flux continues to increase, while the quenching heat flux decreases. The evolution of bubble departure diameter along the heated wall is shown in Figure 7b. The bubble departure diameter increases along the channel due to decreased subcooling and reached maximum value of 0.6 mm at the end of the channel. The wall temperature and saturation temperature are shown in the same figure. Due to pressure drop in a long narrow annulus the saturation and wall temperature decrease of about 4°C over the channel length can be observed. The calculated shear velocity is shown in Figure 7c. It can be noted that the shear velocity increases with evolution of boiling flow.

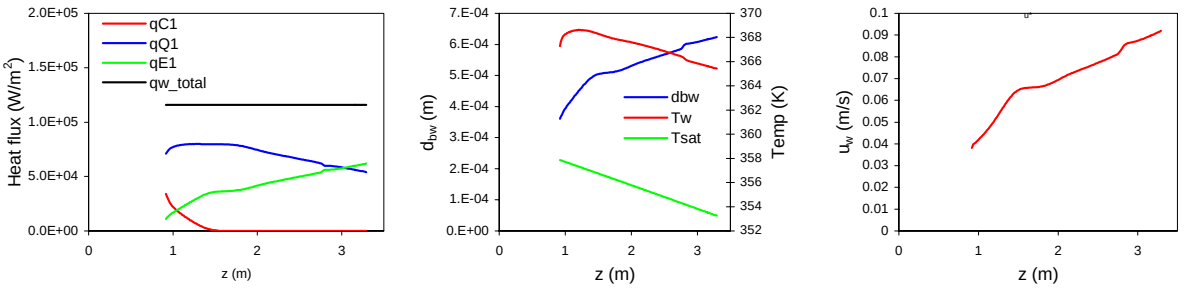

Figure 7 Axial evolutions of wall heat flux components (a), wall temperature and bubble departure diameter (b), shear velocity (c)

5. CONCLUSIONS AND OUTLOOK FOR FUTURE WORK

The capability of the NEPTUNE_CFD V.1.0.6 code to simulate local boiling flow processes has been assessed. Boiling flow experiments from Arizona State University (ASU), Korean Atomic Energy Institute (KAERI) and from DEBORA facility (CEA, France) were used as an experimental benchmark data sets. Experiments differ in geometry, working fluids and boundary conditions.

The sensitivity analysis to the near-wall modelling of velocity has been performed. The use of wall functions is a common approach to overcome the inability of the k–ε model to predict a turbulent velocity profile near the wall. It has been shown that the standard single-phase wall function cannot be straightforwardly applied for prediction of the velocity profile in the boiling boundary layer, which is characterized by a high concentration of bubbles nucleating at the wall. To improve the prediction of phase velocities in the near-wall region a two-phase wall law for momentum equation based on the bubble-equivalent surface roughness has been proposed and implemented into the NEPTUNE_CFD V1.0.6 code. In general, the simulations with the new wall function model show an improved prediction of flow parameters in the boiling boundary layer, but overestimate the turbulent kinetic energy in the boiling region (ASU experiments in annulus). The liquid velocity predictions were improved adjacent to the heated wall, while the values somewhat away from the wall still over-predict experimental data, forming a peaked velocity profile. For the pipe flow (DEBORA), the gas velocity profile shifts from wall-peaked to central-peaked which is in accordance with the measured gas velocity profile. Presented analyses have shown that the implemented velocity wall function gives better predictions of experimental data and contributes to better understanding of wall-to-fluid transfer mechanisms at boiling flow. Still, the remaining experimental needs and outlook for further research are the following:

- Measured values of shear velocity or shear stress in experiments are not known. Experimental data, which would enable reliable estimation of wall shear stress at boiling flows are needed. Beside velocity and temperature profiles, at least pressure drop over the test section should be measured.
- The scaling shear velocity depends on turbulent kinetic energy k_l. This is good for computational purposes, but it may cause over-prediction of k_l due to the feed-back effect. A better approach would be to calculate shear velocity independently from k_l.
- A peaked shape of liquid velocity profile in the near-wall region has been observed when the new wall function model was used. This peak may be affected either by the lift force or by the modelling of the dissipation rate (ε) production term of k-ε equations. Sensitivity analyses of the lift force have shown that the effect of the lift force is negligible. For future research different modelling approaches of ε production term should be analysed to estimate its effect on liquid velocity and turbulent kinetic energy profiles.
- In this study, a formulation of enhanced convective heat flux has been proposed, which takes into account the enhancement of the convective heat transfer due to the increased two-phase wall friction, but it has not yet been incorporated into the CFD code. Near-future work envisages incorporation of this model into the NEPTUNE_CFD code.

ACKNOWLEDGEMENTS

This work was financially supported in part by the Slovenian Research Agency (research project J2-9362-0106-06) and partly through the NURESIM project (6th Euratom Framework Program).

REFERENCES

Bestion, D., Caraghiaur, D., Anglart, H., Péturaud, P., Krepper, E., Prasser, H M., Lucas, D., Andreani M., Smith B., Mazzini D., Moretti F., Macek J., 2006. Deliverable D2.2.1: Review of the Existing Data Basis for the Validation of Models for CHF. NURESIM-SP2-TH D2.2.1.

Garnier, J., François, F., DEBORA data bank (CD), 2006.

Končar, B., Krepper, E., Egorov, Y., 2005a. CFD Modelling of Subcooled Flow Boiling for Nuclear Engineering Applications, International Conference "Nuclear Energy for New Europe" Bled, Slovenia, September 2005, Paper 140

Končar, B., Mavko, B., Hassan Y.A., 2005b. Two-phase wall function for modeling of turbulent boundary layer in subcooled boiling flow. 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France Oct 2005, Paper 443.

Končar, B., Krepper, E., 2007a. CFD simulation of convective flow boiling of refrigerant in a vertical annulus, Nucl. Eng. Des. (In Print), 2007.

Končar, B., Jan. 2007b. Deliverable D2.2.13: Development of physical models for boiling flow and implementation in the NURESIM-CFD code - Modelling of boiling flow boundary layer - simulation of Arizona State University (ASU) boiling flow experiments NURESIM-SP2-TH D2.2.13.1, January 2007.

Kurul, N., Podowski, M.Z., 1990. Multidimensional effects in forced convection subcooled boiling. Proceedings of the Ninth International Heat Transfer Conference, Jerusalem, Israel, August 1990, 21-26.

Lavieville, J., Quemerais, E., Mimouni, S., Boucker, M., Mechitoua, 2005. N.: NEPTUNE CFD V1.0 Theory Manual, EDF 2005.

Lee, T.H., Park, G.C., Lee, D.J., 2002. Local flow characteristics of subcooled boiling flow of water in a vertical concentric annulus, Int. J. Multiphase Flow 28, 1351-1368.

Morel, C., Yao, W., Bestion, D, 2003. Three dimensional modelling of boiling flow for the NEPTUNE code, Proc. NURETH-10, Seoul, Korea, Oct. 5-9, 2003.

Morel, C., Mimouni, S., Laviéville, J.M., Boucker, M., 2005. R113 boiling bubbly flow in an annular geometry simulated with the NEPTUNE code. Proc. NURETH-11, Avignon, France, paper 248.

Ramstorfer, F., Breitschadel, B., Steiner, H., Breee, G., 2005. Modelling of the near-wall liquid velocity field in subcooled boiling flow. Proc. ASME Summer Heat Transfer Conf., San Francisco, CA, July 2005, HT2005-72182.

Ranz, W.E., Marshall, W.R., 1952, Evaporation from drops. Chemical engineering progress 48 (3), 141-146

Roy, R.P., Kang, S., Zarate, J.A., Laporta, A., 2002. Turbulent subcooled boiling flow – experiments and simulations. J. Heat Transfer 124, 73-93.

Troshko, A.A., Hassan, Y.A., 2001. A two-equation turbulence model of turbulent bubbly flows. Int. J. Multiphase Flow 27, 1965-2000.

Unal, H.C., 1976. Maximum bubble diameter, maximum bubble-growth time and bubble growth-rate during the subcooled nucleate flow boiling. Int. J. Heat Mass Transfer 19, 643-649.

Vyskocil, L., Macek, J., 2007. Deliverable D2.2.17.1b: Modification of nucleate boiling model in NEPTUNE. NURESIM-SP2-TH D2.2.17.1b.

Wintterle,T., Eckart, L., Egorov,Y., Menter,F., 2005. On the Customizing of a Subcooled Boiling Model for Industrial CFD Simulation, 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon France Oct 2005, Paper 186.