Simulative approach for two phase flow boiling

Vaibhav T.Patil[1], Prof Sachin P.Komble[2]
[1, 2] Department of Mechanical Engineering
[1, 2] Vishwakarma Institute of Technology, Pune, India

Abstract- *Two phase flow boiling is an important phenomenon in many engineering applications like steam generators, evaporators, microelectronic cooling, nuclear reactors etc. Most of the researchers have carried experimental work for understanding flow boiling behaviours in different applications. Very few researchers have tried flow boiling simulations in different software's. Thus there is a need of developing standard procedures (involving standard models) for flow boiling simulations resulting in more reliable simulated data. This paper provides a brief idea about procedure to develop flow boiling simulation in STAR CCM+ software. An attempt is also made to develop flow boiling simulation in vertical channel using this software.*

Keywords- Flow boiling, STAR CCM+, vertical channels.

I. INTRODUCTION

Two phase flow boiling is important phenomenon in many engineering applications. Experimental studies have concluded many important parameters like flow regimes, heat transfer coefficient and pressure drop. Thus in order to characterize flow boiling in any channels it is necessary to focus on these parameters. Very few researchers have tried simulating flow boiling simulations in order to explain actual physical phenomenon happening in channels

Majority of research in two flow boiling simulations is carried out for nuclear reactors. In the nuclear industry, all water-cooled reactors experience some degree of boiling during normal operation, and boiling flows are key contributors in many design-bases and beyond design-basis postulated accidents. In all such conditions, the critical heat flux (CHF) is an important safety limit, being the maximum amount of heat transferrable from the nuclear fuel to the coolant. Therefore, it is necessary for the CHF in fuel bundles to be accurately predicted in order to avoid overheating and, eventually, melting of the fuel rods

Thus simulating flow boiling in nuclear reactors will help in predicting possible dryout occurring in reactors and preventing the accidents.

M.Colombo et al (2016) have simulated flow boiling inside tubes in nuclear reactors. In this paper the predictive capabilities of a computational fluid dynamic and flow boiling models was tested.

In this work, boiling flows are simulated with the STAR-CCM+ CFD code and an Eulerian-Eulerian two-fluid model in an effort to contribute to the development of more advanced predictive tools for flow boiling experiments. Amongst the CFD approaches, Eulerian-Eulerian averaged two-fluid models are at the present time the only practicable choice when handling industrial-scale engineering problems. In these models, boiling at the wall is incorporated with approaches based in the majority of cases on the Rensselaer Polytechnic Institute (RPI) heat flux partitioning model (Kurul and Podowski, 1990), where the heat flux from the wall is accommodated through single-phase convection, quenching and evaporation. The detailed procedure for two phase flow boiling simulations is explained in sections below.

II. PHYSICS MODELS

Selecting physics model for two phase flow boiling is most important part in Star CCM+ because this model governs the flow throughout the process. The procedure for selecting models is explained in sections below,

Space:

The primary function of the space models in STAR-CCM+ is to provide methods for computing and accessing mesh metrics. Three options are available based on type of mesh we have,

Axisymmetric Model

The Axisymmetric model is designed to work on two-dimensional axisymmetric meshes. When using the model, mesh is oriented such that the axis of rotation is at y = 0 in global coordinate space. No part of the mesh can be below y = 0 and the boundary edge that lies along the axis must be of type axis. These types of models are basically used when your flow is symmetric about the axis as in the case of flow boiling in vertical channels.

Two dimensional Models:

The Two-Dimensional model is designed to work on two-dimensional meshes. In this model, the mesh is assumed to have a unit depth (in SI units) so that any volumetric or area quantities reported for the two-dimensional model are assumed to be "per meter".

Three dimensional Models:

The three dimensional model is used when we have three dimensional mesh.

Selecting a Space Model

To select a Space Model, open the Continua node of the simulation tree, then right-click the Physics 1 node -> Select models -> Space Model
So from this list of models you can select different models based on boiling in channels.
E.g.:

For two phase flow boiling in vertical channels -> Axisymmetric model For two phase flow boiling in horizontal channel -> Two dimensional Model

Time:

The type of time model to be selected depends upon the system you are working on, if the system is steady i.e. the parameters in the experiments remains constants throughout the process, then steady state is best option. The Steady model is used for all steady-state calculations. When this model is activated, the concept of a physical time-step is meaningless.
But if the parameters are changing continuously with respect to time then, unsteady is the best option. In this it is desirable to know the time step for which we are performing experiments. This time step is used for giving total time span for running simulation.

Material

In material node we can select the type of material we are dealing with, for two phase flow boiling the most preferable model is Eulerian multiphase model.

Eulerian - Eulerian Approach:

In the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information, or, in the case of granular flows, by application of kinetic theory.

The Governing equations for Eulerian-Eulerian approach in two phase flows are indicated below,

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{U}_k) = 0 \qquad (1)$$

$$\frac{\partial \alpha_q \rho_q \mathbf{U}_q}{\partial t} + \nabla \cdot (\alpha_q \rho_q \mathbf{U}_q \mathbf{U}_q) = -\alpha_q \nabla P + \alpha_q \nabla \cdot \tau_q + \alpha_q \rho_q \mathbf{g} + \mathbf{S}_q = 0 \qquad (2)$$

$$\frac{\partial \alpha_k}{\partial t} + \nabla \cdot (\alpha_k \mathbf{U}_k) = 0 \qquad (3)$$

Features:

- Used to model droplets or bubbles of secondary phase(s) dispersed in continuous fluid phase (primary phase)
- Allows for mixing and separation of phases.
- Solves momentum, enthalpy, and continuity equations for each phase and tracks volume fractions.
- Uses a single pressure field for all phases.
- Uses interphase drag coefficient.
- Allows for virtual mass effect and lift forces.
- Allows for heat and mass transfer between phases.
- Can solve turbulence equations for each phase

In Eulerian model there are different options involved as follows,

The Multiphase Segregated Flow model: This model is commonly known as the Eulerian Multiphase model. The Multiphase Segregated Fluid model solves conservation equations for mass, momentum, and energy for each phase. Phase interaction models are provided to define the influence that one phase exerts upon another across the interfacial area between them.

The Volume of Fluid (VOF) model: This model is provided for systems containing two or more immiscible fluid phases, where each phase constitutes a large structure within the system (such as typical free surface flows). This approach captures the movement of the interface between the fluid phases, and is often used for marine applications.

The mixture model is designed for two or more phases (fluid or particulate). The mixture model solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. Applications of the mixture model include, bubbly flows, sedimentation, and cyclone separators.

Both the Multiphase Segregated model and the Volume of Fluid model use distinct Eulerian phases.

Viscous Regime:

The type of regime is selected based on velocity of flow. There are two types Laminar and Turbulent.
If the flow is turbulent then we have to select from list of models as given below,

- K-epsilon Turbulence model
- K-omega turbulence model
- Reynolds stress Turbulence model
- Spallart-Allmarus Model

Out of these models Spallart-Allmarus is specially developed for aerodynamic applications, while Reynolds Stress Model is mostly suitable for cyclone flows, highly swirling flows in combustors, rotating flow passages, and the stress-induced secondary flows in ducts. So the remaining two models are explained in detailed in section below,

K-ε Turbulence models:

In the two –equation models, we develop two PDEs one for the turbulent kinetic energy and one for turbulent dissipation rate. In this model the ideal is to express the turbulent viscosity as a function of K and ε and then derive PDEs for K and ε

The expression for K and ε are given below,

Equation for turbulent kinetic energy

$$\frac{\partial \rho K}{\partial t}+\frac{\partial \rho U_j K}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial K}{\partial x_j}\right]+P_k+G_k-\rho\varepsilon \quad (4)$$

$$P_k=\rho\overline{u_i u_j}\frac{\partial \rho U_i}{\partial x_j}=\mu_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} \quad (5)$$

$$G_k=\beta g_i\frac{k_t}{c_p}\frac{\partial T}{\partial x_i} \quad (6)$$

Equation for dissipation rate

$$\frac{\partial \rho \varepsilon}{\partial t}+\frac{\partial \rho U_j \varepsilon}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{\varepsilon 1}\left(P_k+C_{\varepsilon 3}G_k\right)\frac{\varepsilon}{K}-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{K} \quad (7)$$

$$P_k=\rho\overline{u_i u_j}\frac{\partial \rho U_i}{\partial x_j}=\mu_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} \quad (8)$$

The standard values of constants used in this equation are indicated in table below,

Table 1. Standard values of coefficients

C_μ	σ_k	σ_ε	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	$C_{\varepsilon 3}$	Pr_t
0.09	1	1.3	1.44	1.92	0 - 1.0	0.7 – 0.9

K-ω Turbulence models:

In this model the standard k equation is solved, but as a length determining equation ω is used. This quantity is often called specific dissipation from its definition $\omega \propto \varepsilon/k$ The equations for turbulent kinetic energy and specific dissipation rate is given below,

$$\rho\frac{Dk}{Di}=\tau_{ij}\frac{\partial \overline{u_i}}{\partial x_j}-\rho\beta^* f_{\beta'} k\omega+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \quad (9)$$

$$\rho\frac{D\omega}{Dt}=\alpha\frac{\omega}{k}\tau_{ij}\frac{\partial \overline{u_i}}{\partial x_j}-\rho\beta f_\beta\omega^2+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] \quad (10)$$

Optional models:

In optional models select gravity option, using this option flow boiling simulation can be carried out at variable gravity levels. This option is also important during flow boiling in horizontal channels, as in this flow gravity results in causing flow stratification. In this gravity model defining the magnitude and direction is important. In optional model there is option -> Phase Coupled energy solver select this option.

Now click on models, a tree structure will appear which contains a list of models which we have selected, now click on Eulerian phases -> In this option we are going to create two phases that will be present in simulation. Out of the two phases one phase will be liquid while another one will be

gas, after developing the phases define the properties of each phase.

Now in liquid phase select the models based as stated,

1. Liquid
2. Turbulent
3. K- ε
4. Realizable K- ε
5. Constant density
6. High Y+ wall treatment
7. Segregated fluid temperature: The Segregated Fluid Temperature model solves the total energy equation with temperature as the solved variable. Enthalpy is then computed from temperature according to the equation of state.

Now create the gas phase as stated below,

1. Gas
2. Turbulence response

The turbulence response model treats the turbulence of the dispersed phase as an empirical relation to the solution of the turbulence of the continuous phase. The turbulence response coefficient, Ct, defines the correlation between the phases as the ratio of the dispersed phase velocity fluctuation to that of the continuous phase.

$$C_t = \frac{\mu_d'}{\mu_c'} \quad (11)$$

Where:

μ'd is the dispersed phase velocity fluctuation.
μ'c is the continuous phase velocity fluctuation.

Now in order to find out Ct there are different models but for bubbly flow use Issa turbulence model because other models are specifically for gas flow with heavy particles suspended in it.

Issa turbulence model:

The Issa Turbulence Response Model is defined as a correlation for the turbulence response coefficient, Ct, with a volume fraction correction.

$$C_t = 1 + (C_t^* - 1)^{-f(\alpha_d)} \quad (12)$$

Where:

$$f(\alpha_d) = 180\alpha_d - 4.71\times10^3\alpha_d^2 + 4.26\times10^4\alpha_d^3 \quad (13)$$

$$C_t^* = \frac{3+\beta}{1+\beta+2(\rho_d/\rho_c)} \quad (14)$$

$$\beta = \frac{2A_{ij}^D l_e^2}{\alpha_d \mu_c Re_t} \quad (15)$$

$$l_e = C_\mu \frac{k_c^{3/2}}{\varepsilon_c} \quad (16)$$

$$Re_t = \frac{|\mathbf{u}_c'| l_e}{\nu_c} \quad (17)$$

$$|\mathbf{u}_c'| = \sqrt{\frac{2}{3}k_c} \quad (18)$$

Where:
ρd is the dispersed phase density.
ρc is the continuous phase density.
Kc Continuous-phase turbulent kinetic energy,
Cμ is a model coefficient in the Standard K-Epsilon Model.
uc is continuous phase velocity
αd is volume fraction correction for dispersed phase
le=Length scale

In Issa turbulence model the reference phase considered is continuous phase

3. Constant density
4. Segregated fluid temperature
5. S-gamma:

Mostly used for dispersed phase, specify particle size distribution. If there is need include the effects of breakup and coalescence on the predicted size distribution, select the S-Gamma Breakup and S-Gamma Coalescence models in a multiphase interaction that includes the dispersed phase.

Defining the phase interaction:

Multiphase Interaction model, can define the interaction between the liquid and vapor phases.

The most widely used modelling choice for flow boiling of water under a wide range of pressures is the use of the combination of Hibiki-Ishii Nucleation Site Number Density and Kocamustafaogullari Bubble Departure Diameter. The default models (Lemmert-Chawla and Tolubinsky) are much simpler, but can need testing and recalibration for use at different pressures.

Now click on Multiphase interactions -> Phase interaction ->New -> Models

The models used for phase interaction is explained in detail below,

Eulerian Continuous-dispersed phase

Interaction length scale: For continuous-dispersed phase interactions, the interface length scale is taken to be an effective mean diameter of the dispersed phase particles. Where particles are not spherical, this is absorbed as a correction factor into interphase transfer models such as for bubble drag and lift force. This length scale gives a measure of length along which both phases interact.

Interaction area density:

The interaction area density specifies the interfacial area available for drag, heat, and mass transfer between each pair of phases in an interaction. Heat and mass transfer models use the interaction area density directly, while drag models use one quarter of the interfacial area as an estimate of the projected area. Any correction factors, such as for non-spherical particle shapes or particle crowding, are assumed to be covered in correlations for the drag, heat, and mass transfer coefficients.

Turbulent dispersion force:

The effect of turbulence in redistribution of non-uniformities in phase concentration is modelled by an additional turbulent dispersion force in the phase momentum equations.

Interphase mass transfer

The heat transfer from the phase-change interface to each of the two phases on either side of the interface is modelled as:

$$Q_i^{(ij)} = h_i^{(ij)} a_{ij} (T_{ij} - T_i) \quad (19)$$

$$Q_j^{(ij)} = h_j^{(ij)} a_{ij} (T_{ij} - T_j) \quad (20)$$

Where:

Ti and Tj are the bulk temperatures of the continuous and dispersed phases, respectively

Tij is the interface temperature

The interface temperature is often defined as a constant saturation temperature for a particular system pressure.

The heat transfer coefficient for energy supply from the continuous phase is:

$$h_i^{(ij)} = Nu_i k_i / l_{ij} \quad (21)$$

Where,

Nui is the continuous phase Nusselt number. Similarly, for the dispersed phase, the heat transfer coefficient for energy supply is:

$$h_j^{(ij)} = Nu_j k_j / l_{ij} \quad (22)$$

Where,

Nuj is the dispersed phase Nusselt number. In both equations, lij is the interaction length scale for the phase pair usually the bubble size.

The Ranz-Marshall correlation that is used for the continuous phase Nusselt number has the same definition as used for the interphase heat transfer. This correlation is not available for the dispersed phase Nusselt number since convective processes are often poorly defined inside the particle.

The interface mass flux is computed from the heat balance:

$$m^{(ij)} = -\frac{Q_i^{(ij)} + Q_i^{(ij)}}{\Delta h_{ij}} \quad (23)$$

Where:

m(ij) is the mass transfer rate per unit interfacial area for the interaction from phase j to phase i

Δhij is the enthalpy input to create phase j from phase i

Optional models:

Wall boiling:

The most widely used wall boiling model is Kurul and Podowski model, which is stated below. This model is based on heat partition in which heat supplied is partitioned

into three components, convective heat flux, evaporative heat flux and quenching heat flux.

$$\dot{q}''_w = \dot{q}''_{conv} + \dot{q}''_{evap} + \dot{q}''_{quench} \quad (24)$$

These components of heat flux are described below:

q''_{conv} is the convective heat flux ,which describes the removal of heat by single-phase turbulent convection on those parts of the wall that is not affected by boiling. In applications with fixed wall heat flux, this term defines the point at which the wall first exceeds saturation temperature.

q''_{evap} is the evaporative heat flux , which describes the power that is used to produce bubbles from nucleation to departure. This term is a strong function of wall superheat (Tw-Tsat). In fixed heat flux applications, once boiling has started, this term is responsible maintaining a wall temperature that is slightly higher than the saturation temperature.

q''_{quench} is the quenching heat flux , which describes the enhancement of heat transfer, due to the replacement of a departing bubble by an influx of cooler liquid farther away from the wall. Bubble-induced quenching is also known in literature as liquid agitation or pumping. This term is less important when the liquid is close to saturation temperature.

Nucleation site number density:

The original Lemmert Chawla model for nucleation site number density is:

$$\frac{n''}{[m^{-2}]} = (m\Delta T_{sup})^P \quad (25)$$

Where:
n^" = nucleation site density
m = calibration constant, with a default value of 185.0/K
P = superheat exponent, with a default value of 1.805
ΔTsup = wall superheat, which is calculated
as:

$$\Delta T_{sup} = \min(\max(T_{wall} - T_{sat}, 0), \Delta T_{max}) \quad (26)$$

Where,
ΔTmax is the maximum superheat applied to the Lemmert Chawla model

Bubble departure diameter:

There are two standard methods that are implemented for calculating the bubble departure diameter:

Tolubinsky-Kostanchuk:

Correlates bubble departure against liquid subcooling.

Kocamustafaogullari – Correlation that is based on bubble force balance at departure, and calibrated using steam-water data between 0.067 and 141.87 bar.

The Tolubinsky Kostanchuk Departure Diameter model is used with the Lemmert-Chawla Nucleation Site Number Density model.

The Kocamustafaogullari departure diameter model is used with the Hibiki Ishii-Nucleation site number density model.
The default option is to use Tolubinsky Kostanchuk for departure diameter, as appropriate for the default sub-model for nucleation site number density.

The Tolubinsky-Kostanchuk model correlates bubble departure diameter against liquid subcooling

$$d_w = d_0 \exp\left[-\frac{\Delta T_{sub}}{\Delta T_0}\right] \quad (27)$$

Where:
do is the reference diameter with default value 0.0006 m
ΔTo is the reference subcooling with default value 45K
ΔTsub is the subcooling of the liquid next to the wall

$$\Delta T_{sub} = T_{sat} - T_l \quad (28)$$

The minimum and maximum bubble departure diameters can be set with the Minimum Diameter and Maximum Diameter properties respectively. The defaults for these limits are 0.000025 m and 0.0014 m (25 micron and 1.4 mm).

Bubble Departure Frequency:

The standard model that is implemented for bubble departure frequency is the Cole model. This is equivalent to taking a typical bubble rise velocity (estimated using unit drag coefficient) as the velocity scale, over bubble diameter dw as the length scale:

$$f = \sqrt{\frac{4}{3}\frac{g(\rho_f - \rho_g)}{d_w \rho_f}} \quad (29)$$

Where:

g is acceleration due to gravity

ρl is the liquid phase density

Bubble Influence Wall Area Fraction:

When an individual bubble departs from the wall with a diameter dw, subcooled liquid flows in to fill the space underneath the detached bubble. The wall area influenced by this quenching flow is larger than the basic footprint of the bubble.

The standard model that is implemented for the bubble influence wall area fraction follows Kurul Podowski

$$K_{quench} = F_A \frac{\pi d_w^2}{4} n'' \quad (30)$$

Where:

FA = area coefficient for scaling between the nucleation site area density and the wall area fraction the bubble-induced quenching influences.

n^"= nucleation site number density.

The default value for FA is 2.0 and is chosen from experience with the main 45-bar test case in Bartolomei and Chanturiya .

Del Valle Kenning Bubble induced quenching heat transfer:

When a bubble leaves the heated surface, cooler liquid fills the space that it occupied. The heat transfer during this process is known as quenching heat transfer. The Quenching Heat Transfer Coefficient is used to calculate the quenching heat flux.

The Del Valle Kenning model, the quenching heat transfer coefficient is:

$$h_{quench} = 2K_{quench} f \sqrt{\frac{\rho_l C p_l k_l t_w}{\pi}} \quad (31)$$

Where:

Kquench is the bubble influence wall area fraction

f is the bubble departure frequency

ρl is the liquid density

Cpl liquid specific heat

kl is the liquid conductivity

tw is the waiting time between bubble departure and the nucleation of the next bubble:

$$t_w = \frac{C_w}{f} \quad (32)$$

Where:

Cw is the wait coefficient. The default value is 0.8. This value comes from an assumption by Kurul and Podowski that quenching occurs between the departure of one bubble and the nucleation of next. This period is 80% of the departure cycle.

Simulation:

Two phase flow boiling simulation in vertical channel was carried out with R-134a as fluid. The parameters considered during simulation is listed in table below,

Table 2. Parameters used during simulation

Parameter	Value
Channel diameter	4.26 mm
Length	500 mm
Fluid	R-134a
System pressure	8 bar
Inlet velocity	0.3396 m/s
Heat flux	118 kW/m^2
Saturation Temperature	305K

Geometry:

Figure 1. Vertical channel geometry

Thus following the above procedure for flow boiling simulation, the results obtained is indicated below,

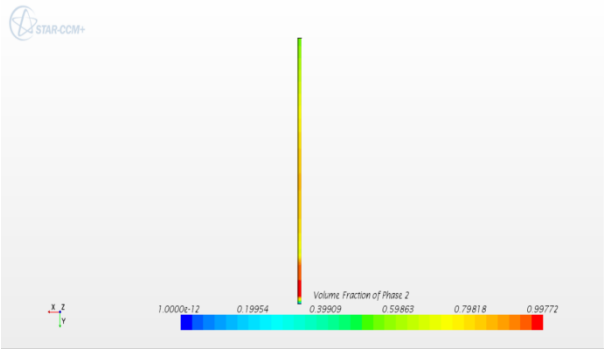

Figure 2. Volume fraction of vapor phase

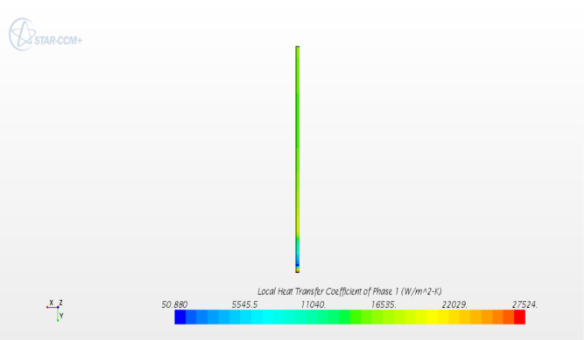

Figure 3. Heat transfer coefficient for Liquid R-134a

The maximum value of heat transfer coefficient obtained is 22000W/m2-K. The red section in Figure 2 indicates complete dryout occurring in channel.

From the simulations carried out it was observed that for higher heat flux and simple geometry the computational power and time required is less but with complex geometries and less heat input it takes time for boiling to occur and hence more computational time and power is required.

III. CONCLUSION

Flow boiling simulations involves many models which has some standard values of coefficients. Variation in results can be obtained by changing these values and running the simulations.

Validation of more experimental results using STAR CCM+ software will help in modifying the standard models to give approximate results.

REFERENCES

[1] CD-adapco, STAR-CCM+, Version 9.04 User Guide.

[2] M. Colombo, M. Fairweather, 2015a, Multiphase turbulence in bubbly flows: RANS simulation, Int. J. Multiphase. Flow 77, 222-243.

[3] M. Colombo, M. Fairweather, 2015b, Modelling of bubbly two-phase flows using a population balance approach, THMT 8, Sarajevo, Bosnia and Herzegovina, 15-18 Sept.

[4] Del Valle, V.H. and Kenning, D.B.R. (1985), Subcooled flow boiling at high heat flux, Int. J. Heat Mass Tran., Vol. 28, pp. 1907-1920.

[5] N. Kurul, M.Z. Podowski, 1990, Multidimensional effects in forced convection subcooled boiling, Proc. 9th Int. Heat Transfer Conf., Jerusalem, Israel, 21-26 Aug.

[6] M. Lemmert, J.M. Chawla, 1977, Influence of flow velocity on surface boiling heat transfer, Academic Press and Hemisphere.

[7] V.I. Tolubinsky, D.M. Kostanchuk, 1970, Vapour bubbles growth rate and heat transfer intensity at subcooled water boiling, Proc. 4th Int. Heat Transfer Conf., Paris, France, 31 Aug-5 Sept.

[8] Fluent 15 User's Guide, September, (2013)