A VERIFICATION AND VALIDATION OF THE NEW IMPLEMENTATION OF SUBCOOLED FLOW BOILING IN A CFD CODE

Francisco A. Braz Filho, Guilherme B. Ribeiro and Alexandre D. Caldeira

Divisão de Energia Nuclear - Instituto de Estudos Avançados
Trevo Coronel Aviador José Alberto Albano do Amarante, 1, Putim
12228-001 São José dos Campos, SP
{fbraz, gbribeiro, alexdc}@ieav.cta.br

ABSTRACT

Subcooled flow boiling in a heated channel occurs when the liquid bulk temperature is lower than the saturation temperature and the wall temperature is higher. FLUENT computational fluid dynamics code uses Eulerian Multiphase Model to analyze this phenomenon. In FLUENT previous versions, the heat transfer correlations and the source terms of the conservation equations were added to the model using User Defined Functions (UDFs). Currently, these models are among the options of the FLUENT without the need to use UDFs. The comparison of the FLUENT calculations with experimental data for the void fraction presented a wide range of variation in the results, with values from satisfactory to poor results. There was the same problem in the previous versions. The fit factors of the FLUENT that control condensation and boiling in the system can be used to improve the results. This study showed a strong need for verification and validation of these calculations, along with a sensitivity analysis of the main parameters.

1. INTRODUCTION

Subcooled flow boiling adjacent to a heated wall is characterized by high heat flux rates. This phenomenon occurs when the liquid bulk temperature is lower than the saturation temperature and the wall temperature is higher. The heat flux in this situation is limited by the critical heat flux (CHF). The CHF causes a sudden increase in the wall temperature that can melt the heating wall.

This study is very important for the thermal hydraulic analysis of Pressurized Water Reactors (PWRs), since CHF is the main design parameter for these reactors.

The Computational Fluid Dynamics (CFD) code, FLUENT [1, 2], uses Eulerian Multiphase Model to analyze the subcooled flow boiling. In FLUENT previous versions, the heat transfer correlations and the source terms of the conservation equations were added to the model using User Defined Functions (UDFs). Currently, these models are among the settings of the FLUENT without the need to use UDFs.

In an earlier article [3], the comparison of the FLUENT previous version results with experimental data was performed. This article shows that the void fraction calculations presented a good agreement, both in the beginning of boiling as in nucleate boiling at the channel outlet. In the region between these two points the agreement with experimental data was not so satisfactory. In the following article [4], a sensitivity analysis of the mass balance equation terms, vapor production and condensation, was carried out to verify the accordance with the experimental results of previous studies.
Recently, an article, published by ANSYS team [5], presented a new implementation of subcooled flow boiling analysis. In that paper, only one case for the pressure of 45 bar was studied. Other researchers have adopted this test case to validate their studies as, for example, Krepper et al. work [6]

The purpose of this work is to perform a comparison of the FLUENT new version calculations with experimental data for pressures of 15, 30 and 45 bar.

2. MODEL DESCRIPTION

Subcooled boiling is observed at heated surfaces, when the heat flux applied to the wall is too high to be transferred to the core flow of liquid by the single-phase convective–conductive mechanisms. The term “subcooled” means that the saturation temperature is exceeded only in a local vicinity of the wall, whereas the bulk temperature is still below saturation. The point where the local wall temperature reaches the saturation temperature is considered as the onset of subcooled boiling. Steam bubbles are generated at the heated surface at nucleation sites. Further downstream the attached bubbles grow and then leave the wall at certain critical size. This critical size may depend on the surface tension and on the flow regime of the surrounding fluid. Heat transfer from the wall is then described as being carried by turbulent convection of liquid, by transient conduction due to the departing bubbles, and by evaporation. Distribution on the entire wall of heat flux between these mechanisms (wall heat partitioning) can be calculated by modeling each mechanism in terms of the nucleation site density, the size of departing bubbles, their detachment frequency, and waiting time until the next bubble appears on the same site (mechanistic modeling approach). When steam bubbles move through the subcooled liquid, they condense, releasing the latent heat.

The model utilized in this work was developed by Kurul and Podowski [7] and has been applied in CFD codes. FLUENT use Eulerian Multiphase Model (EMM) with the addition of heat transfer correlations and source terms in the conservation equations. FLUENT versions have been improved through several works in this area [8-11].

The conservation equations are written for each phase, liquid and vapor, in EMM. The following is a summary of the main equations, for a \( q \) phase.

The description of multiphase flow as interpenetrating continua incorporates the concept of phase volume fractions, denoted here by \( \alpha_q \). Volume fractions represent the space occupied by each phase, and the laws of conservation of mass, momentum and energy are satisfied by each phase individually. The derivation of the conservation equations can be done by ensemble averaging the local instantaneous balance for each of the phases or by using the mixture theory approach.

The volume of \( q \) phase, \( V_q \), is defined by

\[
V_q = \int_V \alpha_q dV ,
\]

where
\[ \sum_{q=1}^{n} \alpha_q = 1 \],

and \( n \) represents the number of phases.

The effective density of \( q \) phase is

\[ \hat{\rho}_q = \alpha_q \rho_q \],

where \( \rho_q \) is the physical density of \( q \) phase.

The mass conservation equation for \( q \) phase is given by

\[ \frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) = \sum_{p=1}^{n} \dot{m}_{pq}, \]

where \( \vec{v}_q \) is the velocity vector and \( \dot{m}_{pq} \) is the volumetric mass exchange rate between \( p \) and \( q \) phases.

The vapor formation rate per unit of volume can be written as

\[ \sum_{p=1}^{n} \dot{m}_{pq} = \dot{m}_v = \frac{(T_l - T_{sat}) H_{R-M} A_i}{L + C_p (T_{sat} - T_l)} + \frac{Q_e A_i}{V_c (L + C_p (T_{sat} - T_l))}, \]

where \( A_i \) is the interfacial area density, \( A_t \) is the interfacial area density of wall, \( \dot{m}_v \) is the vapor formation rate per unit of volume, \( T_l \) is the liquid temperature, \( T_{sat} \) is the saturation temperature, \( V_c \) is the cell volume, \( L \) is the latent heat per unit of mass, \( Q_e \) is the evaporative heat flux, \( C_p \) is the liquid specific heat and \( H_{R-M} \) is the interfacial heat transfer coefficient (Ranz-Marshall).

The momentum conservation equation for \( q \) phase is given by

\[ \frac{\partial}{\partial t} (\alpha_q \rho_q \vec{v}_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q \vec{v}_q) = -\alpha_q \nabla P + \nabla \tau_q + \alpha_q \rho_q \vec{g} + \\
+ \sum_{p=1}^{n} (\tilde{R}_{pq} + \dot{m}_{pq} \vec{v}_{pq}) + (\tilde{F}_q + \tilde{F}_{lif,q} + \tilde{F}_{vm,q}), \]

where \( \vec{F}_q \) is an external body force, \( \tilde{F}_{lif,q} \) is a lift force, \( \tilde{F}_{vm,q} \) is a virtual mass force, \( \tilde{R}_{pq} \) is an interaction force between phases, \( P \) is the pressure shared by all phases, \( \tau_q \) is the \( q \)th phase stress-strain tensor and \( \vec{v}_{pq} \) is the inter-phase velocity. The interfacial drag force per unit of volume is calculated as
\[ \vec{R}_{pq} = \vec{R}_{v} = 0.75 C_d \rho_i \alpha_v |\vec{v}_r| l d_v , \]  

where \( C_d \) is the drag coefficient and \( d_v \) is the bubble diameter.

The energy conservation equation for \( q \) phase is given by

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q h_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q h_q) =
-\alpha_q \frac{\partial p}{\partial t} - \nabla \vec{q}_q + S_q + \sum_{p=1}^{n} (Q_{pq} + \dot{m}_{pq} h_{pq}) ,
\]

where \( h_q \) is the enthalpy, \( \vec{q}_q \) is the heat flux vector, \( S_q \) is the source term, \( Q_{pq} \) is the energy exchange term between the different phases, and \( h_{pq} \) is the difference in the formation enthalpies of \( p \) and \( q \) phases.

According to the Reference 6, part of the total heat flux from wall to liquid phase is partitioned into three components

\[ q_w^* = q_l^* + q_Q^* + q_E^* , \]

which are liquid convective heat flux, quenching heat flux, and evaporative heat flux, respectively. Under subcooled boiling conditions, the wall surface is subdivided into portion \( \phi (0 \leq \phi \leq 1) \), covered by nucleating bubbles, and portion \((1 - \phi)\), covered by fluid. Therefore, convective heat flux is expressed as

\[ q_l^* = h_{lw} \left(T_w - T_l^{cell}\right) \left(1 - \phi\right) , \]

where \( h_{lw} \), the single-phase heat transfer coefficient, is derived from either log law if flow is turbulent or Fourier law if flow is laminar. Liquid phase properties must be used while calculating \( h_{lw} \) for either turbulent or laminar flow.

Quenching heat flux \( (q_Q^*) \) represents additional energy transfer related to liquid filling the wall vicinity after the bubble detachment

\[ q_Q^* = 2\pi^{-0.5} f \kappa_l \rho_l C_l^{0.5} \left(T_w - T_l^{cell}\right) , \]

where \( f \) is the bubble departure frequency, \( \kappa_l \) is the thermal conductivity, \( C_l \) is the specific heat, and \( \rho_l \) is the density.

The evaporative heat flux is given by

\[ q_E^* = \frac{\pi}{6} \alpha_v d_{vw}^3 f n \rho_v L , \]

where \( d_{vw} \) is the bubble departure diameter and \( n \) is the nucleation site density.
The mixture turbulence model is the default multiphase turbulence model and was been applied. It represents the first extension of the single-phase $K-\Omega$ model. In this case, using mixture properties and mixture velocity is sufficient to capture important features of the turbulent flow.

Currently, the source terms of the conservation equations and these heat transfer correlations are implemented in FLUENT without the need to use UDFs.

The problem was solved using a bidimensional and an axisymmetric geometry. The boundary conditions are as follows: i) at the inlet, Velocity Inlet; ii) at the outlet, Pressure Outlet; iii) at the external wall, Heat Flux; iv) at the interface wall-liquid, Coupled; and v) at the centerline axis, Axis. For the pressure-velocity coupling was used Coupled scheme and for the spatial discretization was used First Order Implicit method.

### 3. RESULTS

For the analyzes, experimental data for a heated vertical tube were used [12]. The heated channel consists of a stainless steel tube with an inner diameter of 15.4 mm and length of 2000 mm. The fluid used in the experience was water, flowing in upward direction at the test section. Note that the experimental data of this reference [12] do not have error bars.

The computational model considers two-dimensional axisymmetric geometry. The code manual recommends to use a quadrilateral computational mesh for EMM. As in Li et al.[5], several tests found the independence of the results with the cells quantity. Figure 1 shows that from 10000 to 60000 cells the results were almost identical. However, when the mesh aspect ratio (width/length) was considered different from 1 there was great difficulty in obtaining the convergence of the numerical method.

For all simulated cases in this work the mesh has 20000 cells. Figure 2 presents a piece of the computational mesh near the channel inlet.

![Figure 1. The mesh dependency checking.](image)
Two articles published by the authors [3,4] used the FLUENT-12 [1] version for calculations. In this version, the heat transfer correlations and the source terms of the conservation equations were implemented via UDFs. The K-ξ turbulence model was employed.

In FLUENT-14 [2] version, these formulations are in its settings. Several choices can be made with respect to the interaction models between the phases, liquid and vapor. Li et al.[4] recommend an optimal configuration for the choice of available correlations. Table 1 shows the correlations used in all cases. The recommended turbulence model was SST K-ω for having achieved the best results. Although, the choices of settings have been indicated by Li et al., several simulations corroborated with this better performance.

**Table 1. Choice of correlations.**

<table>
<thead>
<tr>
<th>Model</th>
<th>Eulerian</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boiling model options</td>
<td>RPI boiling model</td>
</tr>
<tr>
<td>Interfacial drag force</td>
<td>Ishii</td>
</tr>
<tr>
<td>Interfacial lift force</td>
<td>Tomiyama</td>
</tr>
<tr>
<td>Turbulent dispersion</td>
<td>Lopez-de-Bertodano</td>
</tr>
<tr>
<td>Turbulent interaction</td>
<td>Troshko-Hassan</td>
</tr>
<tr>
<td>Interfacial heat transfer coefficient</td>
<td>Ranz-Marshall</td>
</tr>
<tr>
<td>Interfacial area</td>
<td>particle</td>
</tr>
<tr>
<td>Bubble departure diameter</td>
<td>Tolubinski-Kostanchuk</td>
</tr>
<tr>
<td>Frequency of bubble departure</td>
<td>Cole</td>
</tr>
<tr>
<td>Nucleation site density</td>
<td>Lemmert-Chawla</td>
</tr>
<tr>
<td>Area influence coefficient</td>
<td>Delvalle-Kenning</td>
</tr>
</tbody>
</table>

The results are divided into three subsections. The first one, called P45_q570, comprises tests with 45 bar pressure and heat flux of 570 kW/m². The second one, P15_30_45q380, consists of tests with pressures of 15, 30, and 45 bar and heat flux of 380 kW/m². The third subsection, P30_45q800, comprises tests with pressures of 30 and 45 bar and heat flux of 800 kW/m².
3.1 P45_q570

Table 2 lists the main input data for this case. This test is particularly important because it presents not only the void fraction data but also the wall temperature, the bulk liquid temperature, and the center line temperature. This case was used in Li et al. [5] to compare FLUENT-14 results with experimental data. They obtained excellent agreement both in relation to temperatures as in void fraction.

Table 2. Input data for pressure of 45 bar.

<table>
<thead>
<tr>
<th>Pressure [bar]</th>
<th>45</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flux [kg/(m(^2) s)]</td>
<td>900</td>
</tr>
<tr>
<td>Heat flux [kW/m(^2)]</td>
<td>570</td>
</tr>
</tbody>
</table>

Figures 3, 4, and 5 show wall temperature, the bulk liquid temperature, and the center line liquid temperature versus flow enthalpy along the channel, respectively. The results present a good agreement with experimental data. The wall temperature in Fig. 3 follows the experimental data, even at the channel end when the temperature slightly decreases. In one-dimensional codes, as COBRA-IV-i and RELAP5 [13], wall temperature have increasing values until the end of the channel.

![Figure 3](image3.png)

**Figure 3. Wall temperature versus flow enthalpy along the channel (45 bar).**

![Figure 4](image4.png)

**Figure 4. Bulk liquid temperature versus flow enthalpy along the channel (45 bar).**
The discrepancies observed in Fig. 6 for the void fraction differ from Li et al. which present a good agreement with the experimental data over all regions. There are two adjustment coefficients which correct the boiling-condensation in the mixture and waiting time between the departure of two consecutive bubbles. The first factor ($f_1$) is applied in the interface heat transfer coefficient between the phases (as Hans-Marshall correlation). The second factor ($f_2$) is applied in the quenching heat flux calculation. Figure 7 shows the void fraction versus flow enthalpy using several values of $f_1$. The lower the factor $f_1$, lower heat exchange occurs between the phases, thus less steam is condensed, causing an increase in the void fraction. Figure 7 presents a good fit of the void fraction with experimental data as $f_1$ decreases. Figure 8 shows the void fraction versus flow enthalpy using several values of $f_2$. Similarly, if $f_2$ decreases, there is less heat exchange due to the detachment of bubbles, so the wall temperature rises causing an increase in the steam production. Both coefficients can be used to adjust the void fraction to the experimental data.
Figure 7. Vapor volume fraction versus flow enthalpy along the channel, using $f_1$ factor (45 bar).

Figure 8. Vapor volume fraction versus flow enthalpy along the channel, using $f_2$ factor (45 bar).

3.2 P15_30_45q380

Table 3 lists the main input data for this case. Figures 9, 10, and 11 present void fraction versus thermodynamic quality along the channel for pressures of 15, 30, and 45 bar, respectively. This test sequence has provided very good results in 45 bar pressure to poor results in pressure of 15 bar, including the prediction of OSB.
Table 3. Input data for pressures of 15, 30 and 45 bar.

<table>
<thead>
<tr>
<th>Pressure [bar]</th>
<th>15, 30, 45</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flux [kg/(m² s)]</td>
<td>900</td>
</tr>
<tr>
<td>Heat flux [kW/m²]</td>
<td>380</td>
</tr>
</tbody>
</table>

Figure 9. Vapor volume fraction versus thermodynamic quality (15 bar).

Figure 10. Vapor volume fraction versus thermodynamic quality (30 bar).

Figure 11. Vapor volume fraction versus thermodynamic quality (45 bar).
3.3 P30_45q800

Table 4 lists the main input data for this case. Figures 12 and 13 present void fraction versus thermodynamic quality along the channel for pressures of 30 and 45 bar, respectively. These tests show poor results compared to experimental data. They underestimate the void fraction values. Apparently, comparing the results for heat flux of 380, 570, and 800 kW/m², the model moves away from the experimental data with the increased heat flux.

Table 4. Input data for pressures of 30 and 45 bar.

<table>
<thead>
<tr>
<th>Pressure [bar]</th>
<th>30, 45</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flux [kg/(m² s)]</td>
<td>900</td>
</tr>
<tr>
<td>Heat flux [kW/m²]</td>
<td>800</td>
</tr>
</tbody>
</table>

![Figure 12. Vapor volume fraction versus thermodynamic quality (30 bar).](image)

![Figure 13. Vapor volume fraction versus thermodynamic quality (45 bar).](image)

4. COMMENTS AND CONCLUSIONS

FLUENT CFD code uses the Eulerian Multiphase Model to analyze the subcooled flow boiling phenomenon. There is the need to complete the model with the implementation of heat transfer correlations and source terms in the conservation equations. Currently, these models are among the settings of the FLUENT code.
The main objective of this work is the verification and validation of FLUENT code calculations for subcooled flow boiling analysis. The tests show results from satisfactory quality values to poor results. Although, the fit factors of the FLUENT code that control condensation and boiling in the model can be used to improve the results.

The results indicate a better agreement with experimental data in the cases of lower heat flux and higher pressures.

This study showed that there is still a need for further comparisons of FLUENT code calculations with experimental data and a sensitivity analysis of the main parameters.

REFERENCES

2. ANSYS, "ANSYS FLUENT 14.5 Documentation", 2013.