Numerical Simulation and Control of Two-Phase Flow with Evaporation in a Vertical Tube Submitted to a Conjugate Heat Transfer

Ghazali Mébarki and Samir Rahal
Department of Mechanical Engineering, LESEI Laboratory, Faculty of Technology, University Hadj Lakhdar of Batna, Batna 05000, Algeria

Received: August 07, 2012 / Accepted: December 04, 2012 / Published: July 31, 2013.

Abstract: A better understanding of two-phase flows with evaporation allows leading to an optimal design of evaporators. For that purpose, numerical simulations are very useful. In this paper, a numerical study has been carried out in order to model and simulate the combination of a two-phase flow with evaporation in a vertical tube. The VOF (volume-of-fluid) multiphase flow method and a phase-change model for the mass transfer have been used. For an accurate modeling, the effect of axial conduction has been also taken into account using a conjugate heat transfer model. Since thermal oscillations are undesirable as they can lead to the failure of the tube, flow instabilities have also been analyzed, using FFT (fast Fourier transforms), in order to comprehend their behavior and influence. A control study of the flow instabilities in the tube is also presented. For that purpose tube inlet temperature has been varied using a gain control parameter.

Key words: Two-phase flow, evaporation, VOF method, conjugate heat transfer, flow instabilities, control of instabilities.

1. Introduction

Two-phase flows with evaporation are involved in many industrial systems such as power generation and refrigeration. A better understanding of these flows allows leading to an optimal design of heat exchangers particularly evaporators. Such design allows reducing manufacturing cost and energy consumption in order to obtain better performances [1]. Two-phase flows with phase change are known to be very complex. Indeed, in addition to classical effects such as buoyancy, viscosity and pressure forces involved in single-phase flows, two-phase flows are also influenced by interfacial tension forces and the exchange of momentum between the liquid and vapor phases [2]. Numerical simulations are widely used to investigate two phase flows with evaporation. For that purpose, several methods have been proposed to model two phase flow such as the VOF (volume-of-fluid) method [3]. In order to model evaporation, appropriate source terms need to be added in both momentum and energy equations to take into account heat absorption during evaporation or heat release during condensation. For example, numerical simulations have been carried out by Ref. [1] to investigate a refrigerant flow boiling in a horizontal serpentine tube using a phase-change model for the mass transfer. The numerical simulation procedure has been validated by an experimental study. The comparison of the numerical and experimental results showed a fair agreement and allowed to reach very interesting features and conclusions. Indeed, the numerical results allowed explaining accurately the bend effects on the flow regimes and the
non-equilibrium thermal release observed in experiments. More recently, Alizadehdakhel et al. [4] conducted a study of two-phase flows with simultaneous evaporation and condensation phenomena in a thermosiphon. Using the VOF method, temperature profiles obtained numerically were compared to experimental measurements and a good agreement has been observed. Thanks to the study of Ref. [4], it has been concluded that numerical simulations are very useful to model and study the complex flow and heat transfer in a thermosiphon.

Two-phase flow instabilities have been observed in many industrial fields such as refrigeration systems and two-phase flow heat exchangers. Recently, there have been many investigations devoted to two-phase flow instabilities [5-8]. For example, Comakli et al. [5] has conducted experimental investigation of two phase flow instabilities in tubes. It has been found that the flow is less stable for larger inlet temperatures. Another conclusion reached is that the periods and amplitudes of both pressure drop and density wave oscillations decrease with the inlet temperature.

The instabilities caused by two-phase flow instabilities with boiling can be classified as either static instabilities which are induced by a steady-state boiling process or dynamic instabilities initiated by transient changes in the boiling process. Dynamic instabilities include mainly density-wave oscillations, pressure drop and thermal oscillations. Thermal oscillations induced by flow instabilities can be particularly detrimental because they can lead to the failure of tubes caused by a continuous cycling of a wall temperature [9, 10]. In Ref. [9], it has been confirmed that under certain conditions, failures in tubes can occur due to the increase in wall temperature and also because of the thermal fatigue resulting from a continuous cycling of the wall temperature. Liang et al. [11] investigated experimentally the static and dynamic instabilities of two-phase flow in a horizontal straight tube evaporator of a refrigeration system. It has been showed that density-wave oscillations occur at most velocities. The oscillation period is about one to three seconds. On the other hand, pressure drop oscillations take place in a negative slope area with a period of about 10 seconds. As for the thermal oscillations, they can be induced by large flow velocities with periods of 60 s and amplitudes which are the largest among the three oscillations. These results can then be used to provide guidance for design and control strategy of refrigeration systems in order to avoid or at least control oscillation problems.

From this brief review, it has been shown that flow instabilities must be more understood in order to control them efficiently. At first, the numerical simulation procedure is described below. Secondly, a validation study has been carried out; some typical results are also displayed and discussed. At the end of the paper, some conclusions are given.

2. Numerical Simulation Procedure

2.1 The VOF Model Theory

The modeling of the two phase flow evaporation was achieved using the VOF (volume of fluid) method, with an UDF (user defined function) as an evaporation model within the FLUENT software. The VOF model can be used to model two or more immiscible fluids by solving a single set of transport equations and tracking the volume fraction of each fluid throughout the numerical cell. Appropriate jump boundary conditions are imposed at the interface. The only drawback of the VOF method is the so-called artificial (numerical) coalescence of gas bubbles which occurs when their mutual distances are less than the size of the computational cell. This fact makes also this approach memory consuming for the simulation of dispersed multiphase flows such as in large systems [12].

The vapour-liquid interface is tracked thanks to the distribution of the vapor volume fraction ($\varepsilon_g$) in a computational cell with $\varepsilon_g = 0$ in the liquid phase and $\varepsilon_g = 1$ in the vapor phase. Therefore, there is a vapor–liquid interface in the cell for $\varepsilon_g$ varying from 0 to 1. During the numerical simulations, the
geo-reconstruct (geometric reconstruction scheme), that is based on the PLIC (piecewise linear interface calculation) method, is applied to reconstruct the bubble free surface. The first step in this reconstruction scheme is the position calculation of the linear interface relative to the centre of each partially filled cell, which is based on information concerning the volume fraction and its derivatives in the cell. The second step is the fluid advection amount calculation through each interface using the computed linear interface representation and information about the normal and tangential velocity distribution at the interface. As a last step, the volume fraction in each cell is determined using the balance of mass flow rates calculated previously [13].

2.2 Governing Equations

The following governing equations are solved simultaneously:

Continuity:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho u)}{\partial y} + \frac{\partial (\rho u)}{\partial z} = S_M
\]  

Momentum:
\[
\frac{\partial (\rho u)}{\partial t} + \sum_{j=1}^{3} \frac{\partial (\rho u u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \sum_{j=1}^{3} \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_j}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial}{\partial x_j} \right) \right) + S_{F,j}
\]  

Energy:
\[
\frac{\partial (\rho E)}{\partial t} + \frac{\partial (\rho E u_j)}{\partial x_j} = \sum_{j=1}^{3} \frac{\partial}{\partial x_j} \left( \tau_{eq} \nu_j \right) - \sum_{j=1}^{3} \frac{\partial}{\partial x_j} q_j^f + S_E
\]

In these equations, the density (\(\rho\)), dynamic viscosity (\(\mu\)) and thermal conductivity (\(k\)) of the fluid depend on the volume fractions of each phase and they are calculated using the following equations:
\[
\rho = \varepsilon_l \rho_l + \varepsilon_g \rho_g
\]
\[
\mu = \varepsilon_l \mu_l + \varepsilon_g \mu_g
\]
\[
k = \varepsilon_l k_l + \varepsilon_g k_g
\]

In the VOF model, \(E\) (energy) is considered as a mass-averaged variable as follows:
\[
E = \sum_{k=1}^{2} \varepsilon_k \rho_k E_k / \sum_{k=1}^{2} \varepsilon_k \rho_k
\]  

where, \(k = 1\) for liquid and \(k = 2\) for vapor.
\[
E_l = C_{r,l} (T - T_{sat}) \quad E_v = C_{r,v} (T - T_{sat})
\]  

The tracking of the interface between phases has been accomplished by solving continuity equations for the volume fractions of the two phases, using the following equation:
\[
\frac{\partial \varepsilon_l}{\partial t} + \mu \nabla \varepsilon = 0
\]

2.3 Evaporation Model

The working liquid (water in our case) evaporation, within the fluent software, was modeled by developing an UDF, for which source terms have been added in the continuity and energy equations. For the evaporation process, equations proposed by De Schepper et al. [14] were used to calculate the source terms (the mass transfer from liquid to vapor and that from vapor to liquid) as follows:
\[
S_{M,l\rightarrow g} = -t_{lg} \varepsilon_l \rho_l \frac{T_l - T_{sat}}{T_{sat}} \quad \text{if } T_l \geq T_{sat}
\]
\[
S_{M,g\rightarrow l} = t_{lg} \varepsilon_g \rho_g \frac{T_{sat} - T_g}{T_{sat}} \quad \text{if } T_g \leq T_{sat}
\]
\[
S_E = -t_{lg} \varepsilon_l \rho_l \frac{T_l - T_{sat} \Delta H}{T_{sat}}
\]

Phase change (evaporation) in the tube was considered to occur at the boiling temperature \(T_{sat}\). When the liquid phase temperature exceeds the saturation temperature, \(T_{sat}\), mass with a \(S_M\) rate transfers from the liquid phase to the vapor one and an amount of energy is then absorbed. In order to avoid convergence problems and to keep the interface temperature within \(T_{sat} \pm 1\) K, the mass transfer time parameters \(t_{lg}\) and \(t_{lg}\) have been taken equal to 0.1 s\(^{-1}\). No momentum exchange between the phases is assumed. Consequently, the source term \(S_{F,j}\) in the momentum equation is 0 [13].

The surface tension of water in contact with its vapor has been introduced using the commonly used equation
which is given with great accuracy in m·N/m:

\[ \sigma = 235.8 \left(1 - \frac{T_{sat}}{647.096}\right)^{1.256} \left[1 - 0.625 \left(1 - \frac{T_{sat}}{647.096}\right)\right] \quad (13) \]

2.4 Resolution Procedure

For all our calculations, liquid was defined as the primary phase and the vapour as the secondary one in the multiphase VOF model. After the UDF compilation, source terms were added in the momentum and energy equations. The SIMPLE algorithm has been used for the resolution of the pressure-velocity coupling. As for the discretization of momentum and energy equations, a second order accuracy upwind scheme has been chosen. For the volume fraction calculation, the “geo-reconstruct” method has been used. To model turbulence, the RNG \( k-\varepsilon \) model has been chosen, this model is more accurate and reliable than the standard \( k-\varepsilon \) one. Conjugate conduction-convection heat transfer at the fluid-solid interface has also been considered by taking into account the tube wall thickness. In this case, both heat flux and temperature at the interface are unknown; they are determined by simultaneous and coupled solutions of the energy equation in the solid and the thermo-fluid equations in the fluid. Since the evaporating two-phase flow is known to have a dynamic behaviour, unsteady state calculations, with a 0.01 s time step, were conducted. Some of the obtained results which are presented in this paper are for a relatively short tube made of cooper or aluminium with a 500 mm length and a 16 mm diameter (Fig. 1). The thickness of the tube wall is 1 mm. 100 kW/m² or 350 kW/m² heat fluxes have been applied. The tube inlet conditions are as follows: \( v = 0.03 \) or 0.3 m/s (Reynolds number \( Re = 480 \) or 4,800 corresponding respectively to laminar or turbulent regimes), \( t = 370 \) K and \( p = 2 \) bars.

The tube geometry has been generated using the gambit software, with refined grids near the walls, as shown in Fig. 2. A total number of 10,000 elements were used for the solid wall and 150,000 elements have been generated in the fluid zone.

3. Results and Discussion

3.1 Validation of the Numerical Procedure

In order to validate our numerical simulation procedure, at first the two phase flow and boiling regimes obtained, using our numerical simulations, have been compared to those obtained by another author [16]. It can be noticed that there is a similarity of the void fraction shapes between the void fractions contours determined using our approach (Fig. 3) and the flow...
Numerical Simulation and Control of Two-Phase Flow with Evaporation in a Vertical Tube Submitted to a Conjugate Heat Transfer

regime chart obtained by Ref. [16], which is shown in Fig. 4. Indeed, there is a fair qualitative agreement. For both figures, in the lowest region of the tube, the liquid phase is more important, while for the highest region, the inverse phenomenon is observed (i.e., vapor phase more important).

During this validation study, the vapour fractions in the tube as well as heat transfer rates, calculated using our numerical procedure, have also been compared to that obtained by other authors. Concerning the heat transfer rates, the presence of both nucleate and convective boiling terms has been considered in the correlation introduced by Kandlikar and Steinke [17] and Kandlikar and Balasubramanian [18] as follows:

For liquid only Reynolds number \( Re_{LO} > 100 \):

\[
h_{TP} = \max(h_{TP,NBD}, h_{TP,CBD})
\]

with:

\[
h_{TP,NBD} = 0.6683Co^{-0.2}(1-x)^{0.8}h_{LO}
\]

\[
+1058Bo^{0.7}(1-x)^{0.8}F_{Fr}h_{LO}
\]

is the two-phase flow boiling heat transfer coefficient.

And:

\[
h_{TP,CBD} = 1.136Co^{-0.8}(1-x)^{0.8}h_{LO} + 667.2Bo^{0.7}(1-x)^{0.8}F_{Fr}h_{LO}
\]

being the two-phase convective heat transfer coefficient.

\[
Co = \left[ \frac{(1-x)/x} \right]^{0.8}(\rho_g/\rho_l)^{0.5}, \quad \text{is the convection number and:}
\]

\[
Bo = q^*/\bar{m}h_g, \quad \text{being the boiling number.}
\]

\( x \) is the mass quality, \( \rho_g \) and \( \rho_l \) are respectively the vapor and liquid density, \( q^* \) is the applied wall heat flux, \( \bar{m} \) is the mass flux and \( h_g \) is the vaporisation latent heat.

For the single phase all-liquid flow, heat transfer coefficient, \( h_{LO} \), Gnielinski [19] correlation has been used, as follows:

For \( 3,000 \leq Re_{LO} \leq 10^4 \):

\[
h_{LO} = \frac{Re_{LO}^{2/3}Pr(1/3)(\rho_l/\rho_g)^{1/3}}{1 + 12.7(Pr^{2/3} - 1)(Re_{LO}/1000)^{1/3}}
\]

For \( 100 \leq Re_{LO} \leq 1,600 \):

\[
h_{LO} = \frac{Nu_{Bo}k_l}{D}
\]

In the transition region (Reynolds numbers from 1,600 to 3,000) a linear interpolation is suggested for \( h_{LO} \). For the fluid surface parameter (\( F_{Fr} \)), it has been taken equal to 1 for water.

In our numerical simulation procedure, the heat transfer rate has been calculated, as follows:

\[
h = \frac{q^*}{T_w - T_b}
\]

The fluid bulk temperature being defined as:

\[
T_b = \frac{1}{A_u \mu_w} \int uT dA
\]
Numerical Simulation and Control of Two-Phase Flow with Evaporation in a Vertical Tube Submitted to a Conjugate Heat Transfer

\( u_m \) is the mean velocity of the fluid mixture, \( u \) and \( T \) are respectively the axial velocity and temperature profiles along the tube.

The tube has been divided into several sections and at each of them, \( q_w^* \), \( T_w \), \( T_b \) and \( h \) have been calculated.

In Fig. 5, the heat transfer rate calculated using our numerical simulation is shown as a function of the position. The heat transfer rate obtained by the Kandlikar Eqs. (14)-(18) is also displayed. It should be noticed that in Eqs. (15) and (16), the introduced values of \( (x) \) have been calculated using our numerical procedure. It is obvious that a very fair agreement has been reached between the two approaches especially for higher positions along the tube for which the flow regime is well established and disturbances due to the entrance effects disappear.

Accurate calculations of the void fraction are also essential in order to understand the two-phase flow phenomena. For that purpose, void fractions have been calculated using our numerical procedure and compared to those obtained by other authors [20].

In Ref. [20], void fractions have been obtained using experimental data combined with various correlations such as Refs. [21, 22] which can be used for any flow regime, pressure or mass flux. Those correlations are given below:

Chen correlation:

\[
\varepsilon = \left( 1 + 0.18 \left( \frac{1-x}{x} \right)^{0.6} \left( \frac{\rho_g}{\rho} \right)^{0.33} \left( \frac{\mu_g}{\mu} \right)^{0.07} \right)^{-1}
\]  

(21)

Hamersma & Hart correlation:

\[
\varepsilon = \left( 1 + 0.26 \left( \frac{1-x}{x} \right)^{0.67} \left( \frac{\rho_g}{\rho_l} \right)^{0.33} \right)^{-1}
\]

(22)

In Fig. 6, the void fraction calculated using our numerical procedure is shown as a function of the position for \( Re = 480 \) and an imposed heat flux \( q_w^* = 100 \text{ kW/m}^2 \). The void fractions obtained using respectively Refs. [21, 22] are also displayed. As mentioned above, for the calculations of \( (\varepsilon) \) using Eqs. (21) and (22), the introduced values of \( (x) \) have been calculated using our numerical procedure. It can be noticed that there is a qualitative agreement between our numerical results and those obtained by the two correlations. Indeed, the same general tendency is observed for the results obtained by all approaches. The void fraction values, calculated using our numerical approach, are closer to those obtained by the Chen et al. correlation for the lowest part of the tube. For the highest part of the tube, the numerically obtained values of the void fraction are better fitted by the Hamersma et al. correlation which does not take into account the ratio of the dynamic viscosity of both phases (i.e., liquid and vapor).

Thanks to the results of the comparison study shown above (Figs. 3-6), our numerical simulation procedure has been validated and consequently a further detailed
and parametric study may be carried out. Indeed numerical results obtained by our procedure might be very useful instead of parametric experimental studies which are costly and more time consuming.

3.2 Flow Instabilities

In Fig. 7, the temperature, at y = 50 mm, as a function of time for two values of the applied lateral heat flux, is shown for the same value of Re (= 480) corresponding to a laminar flow. It can be noticed that, as expected, larger values of temperature are obtained for the larger value of the heat flux (= 350 kW/m²). Moreover, larger temperature oscillations, with magnitude increasing as a function of time, are observed for \( q_w = 350 \) kW/m² compared to those for the lower value of \( q_w = 100 \) kW/m². Void fraction as a function of time is also shown in Fig. 8 for the same conditions as Fig. 7. It is obvious that the larger heat flux (= 350 kW/m²) allows to reach larger values of the void fraction (around 0.9) even for a small y (= 50 mm) and a short period of time (t = 8 s). Nevertheless and as mentioned above, large temperature oscillations need to be avoided for the evaporation systems, we will then restrict ourselves to \( q_w = 100 \) kW/m² in the numerical study carried out below.

Fig. 9 shows the development of the vapor phase along the tube for both laminar and turbulent regimes. It is obvious and as expected, the phase vapour takes place at first near the heated lateral wall while the liquid phase is located in the central part of the tube with its amount decreasing with y. The rate of decrease depends on the flow regime (laminar or turbulent). In Fig. 10, time variation of the void fraction, at three positions along the tube, are shown. The appearance of the phase vapor occurs after 1 s for all positions but further time variation depends on the considered position. It can be noticed that void fraction oscillations increase with position (from 50 mm to 450 mm).

The influence of the Reynolds number on the time variations of the void fraction is shown in Fig. 11. It is obvious that for the Re = 480, corresponding to a
Numerical Simulation and Control of Two-Phase Flow with Evaporation in a Vertical Tube Submitted to a Conjugate Heat Transfer

Fig. 10 Time variation of the void fraction at various positions along the tube wall. \( Re = 4,800 \), Heat flux = 100 kW/m\(^2\), tube made of aluminum.

Fig. 11 Void fraction as a function of time for two values of the Reynolds number. \( y = 450 \) mm, from the inlet, Heat flux = 100 kW/m\(^2\), tube made of aluminum.

In the laminar regime, the void fraction tends to reach the maximal value for the void fraction (≈ 1) while the maximal value of the void fraction for the turbulent regime (\( Re = 4,800 \)) fluctuates around 0.4. It can also be noticed that the void oscillation magnitudes are lower compared to those obtained for the turbulent regime corresponding to \( Re = 4,800 \).

Temperature fields at \( t = 10 \) s, corresponding respectively to laminar and turbulent regimes, are shown, in Fig. 12. In Fig. 13, the heat transfer rate \( h \) as a function of the position \( y \) is displayed for two values of the Reynolds number. Heat transfer rates values have been calculated using the temperature fields and the Eqs. (18) and (19) described above. It can be noticed that the turbulent regime, corresponding to \( Re = 4,800 \), allows reaching larger values of \( h \) compared to heat rates obtained for \( Re = 480 \), corresponding to a laminar regime.

Fig. 12 Contour plots of temperature at \( t = 10 \) s. Heat flux = 100 kW/m\(^2\), tube made of aluminum. (a): \( Re = 4,800 \), (b): \( Re = 480 \).

Fig. 13 Heat transfer coefficient as a function of the position along the tube. Heat flux = 100 kW/m\(^2\), tube made of aluminum.

As mentioned above, temperature oscillations need to be studied in order to be avoided or at least be controlled. For that purpose, the temperature time variations for various positions are shown in Fig. 14 for \( Re = 4,800 \) corresponding to the turbulent regime for
Numerical Simulation and Control of Two-Phase Flow with Evaporation in a Vertical Tube Submitted to a Conjugate Heat Transfer

which higher values of heat transfer rates have been obtained (Fig. 13).

As mentioned above and the same way as for the void fraction, the oscillations depend on the considered position and they are more important for higher positions (250 mm and 450 mm) than for the lower one (50 mm). The FFT (fast Fourier transforms), corresponding to the curves of Fig. 14, are displayed in Fig. 15. It can be noticed that for all positions, the spectra are continuous and higher magnitudes are observed for the lower values of the frequency.

3.3 Control of the Flow Instabilities

As mentioned above, thermal oscillations are detrimental to the tube and the goal of our work is to understand their behaviour in order to find the best means to control them. In this paper a control approach is proposed, which is the control of the inlet sub-cooled temperature, in a way similar to that used by Kechil and Hashim [23], who introduced a gain control parameter named \( G_c \) which has been varied and its effect on the oscillations studied.

In this study, in the inlet temperature, a gain control parameter \( G_c \) has been included as follows:

\[
T_{in} = T_i + G_c T_2
\]  

With \( T_i = 354.5 \) K corresponding to the saturation temperature at \( P = 0.5 \) bar and \( T_2 = 400.5 - 354.5 = 46 \) K, the value 400.5 being the saturation temperature at \( P = 2.5 \) bar.

In our simulations, the inlet pressure was taken between the two saturation pressures (0.5 and 2.5 bar) and \( G_c \) was varied from 0.23 to 0.43 corresponding to \( T_{in} \) from 356 K to 374 K. As for the value of \( G_c = 0.33 \), it is the reference value and corresponds to the case of the flow without control.

Temperature and void fraction as functions of time are shown in Figs. 16 and 17 for various values of \( (G_c) \) for \( Re = 480 \) corresponding to the laminar regime for which it has been found that the void fraction reaches the largest values with smaller oscillations (Fig. 11). It is obvious that the larger value of \( (G_c) = 0.43 \) allows to obtain larger void fraction values with less oscillations.

4. Conclusions

In this paper, a numerical study has been carried out in order to model and simulate the combination of a two-phase flow with evaporation in a vertical tube. For that purpose, the VOF multiphase flow model and a phase-change model for the mass transfer have been used. Temperature and void fraction fields have been obtained. The authors believe that the numerical procedure proposed in this paper will be very useful in designing efficient evaporation systems.

In this study, it has been found that the laminar regime allows obtaining larger values of the void fractions with less oscillations although heat transfer
rates for a turbulent regime are much larger than for the laminar regime.

Since thermal oscillations are undesirable as they can lead to the failure of the tube, flow instabilities have been analyzed in order to comprehend their behaviour and influence. A parametric control study of the flow instabilities in the tube has also been presented. For that purpose, tube inlet temperature has been varied and the best values of the gain control parameter determined.

Acknowledgment

The authors are profoundly grateful to the Ministry of Higher Education and Scientific Research, of People’s Democratic Republic of Algeria for financial support of this research PNR project (National Research Project) with contract Nr. 31/06.

References

Numerical Simulation and Control of Two-Phase Flow with Evaporation in a Vertical Tube Submitted to a Conjugate Heat Transfer


