Analysis of water condensation and two-phase flow in a channel relevant for plate heat exchangers

J. Yuan, C. Wilhelmsson & B. Sundén
Department of Energy Sciences, Faculty of Engineering, Lund University, Sweden

Abstract

Water vapor condensation and two-phase flow appear in plate heat exchangers being used as condensers. Analysis of water phase change and flow dynamics is an important but complicated task due to large change in water physical/transport properties across the water liquid-vapor interface boundary. In particular, a singular-link behaviour in Navier-Stokes (N-S) equations is present due to the large step change in the density when computational simulation methods are applied. Conventional methods using ensemble averaged parameters such as void fraction cannot be applied to cases where high-resolution calculations and detailed analysis are required. In this study, a computational fluid dynamics (CFD) approach is presented for analysis of water vapor condensation and two-phase flow in a channel relevant for plate heat exchanger parallel plates. The developed model is based on the governing equations which are directly solved for the entire single- and two-phase fields. The water phase change and two-phase flow are treated by employing a water liquid-phase fraction factor based on the total enthalpy in each computational cell. The factor is defined as the ratio of the total enthalpy differential to the latent heat of condensation. The density, viscosity and conductivity of the two-phase region are calculated and updated based on the calculated value of the liquid-phase fraction factor until a converged result is reached. It is revealed that, among others, the inlet vapor velocity has significant effects on the water phase change and two-phase flow in the channel, in terms of liquid-water fraction factor distribution.
Keywords: water, condensation, two-phase flow, model, analysis.

WIT Transactions on Engineering Sciences, Vol 53, © 2006 WIT Press
www.witpress.com, ISSN 1743-3533 (on-line)
doi:10.2495/HT060341

1 Introduction

It is clear that new cooling strategies are required for many applications such as electronic devices because they are becoming more compact and rejected heat flux is increasing. Condensation and boiling occur in compact heat exchangers, e.g., plate heat exchanges (PHEs), which can remove larger amounts of heat. In these cases, it is necessary to gain a fundamental understanding of phase change and two-phase flow phenomena in PHE channels [1].

The traditional concept of a PHE is the plate-and-frame heat exchanger, which consists of the plates, the gaskets, the frames and some additional components (the carrying and guiding bars, the support column, etc.). The two streams flow into alternate channels between plates, entering and leaving via ports at the corner of the plates. A stream exchanges heat with the streams in adjacent channels. Because of their structures, PHEs offer a number of advantages over conventional shell-and-tube heat exchangers. These advantages are compactness, high effectiveness, flexibility, easy cleaning, cost competitiveness, etc. PHEs have been used for condensation applications for ammonia, ethanol, hydrocarbons, water vapor, etc. In general, the gas enters at the top of the plate and moves downwards over the heat exchanging surface. The condensate leaves at the bottom, while the cooling medium is flowing in neighbouring channels either in counter flow or parallel flow. Water vapor condensation is one of the most common two-phase applications for PHEs [1].

In two-phase applications, the major efforts have been made in the field of experimental investigations with phase change. Due to the small corrugated channels, the flows and heat transfer in PHEs are complicated, and relevant models are very rare.

2 Problem statement

It is known that the condensation performance is dependent on many factors, such as fluid properties, plate geometry, system pressure, mass flow rate, etc. The water vapor condensates because the plate temperature is below the water vapor saturation temperature, and consequently forms a thin film. The latent heat released during condensation must be transferred to the plate through the conduction of the condensate film. Heat transfer during the condensation from the liquid-vapor region is very important for predicting performance of two-phase PHEs. In all heat transfer problems involving liquid-vapor phase change, there is a sharp jump in the value of the density and other physical/transport properties across the liquid-vapor interfacial boundary. This sudden change in the properties across the interface challenges the convergence of all traditional numerical schemes that have been successful for single-phase systems [2–6]. There are several CFD algorithms developed during last decades to simulate two-phase flow processes, such as the two-fluid model, the step-function model and the volume of fluid (VOF) method, as summarized in table 1 [7–9].

Table 1: Various CFD models for two-phase flow [7–9].

Model	Features	Remarks
Two-fluid model	Using two sets of phase N-S equations plus a set of empirical relationships for the interfacial mass transport and wall to fluid heat transport.	Very limited to the cases within the range of the experimental correlations.
Step-function model	Using a set of N-S equation for both phases plus a set of step-functions for the liquid, vapor and two-phase regions.	Very often leading to prediction of unrealistic interface.
Volume of fluid (VOF) model	Directly simulating and tracking of the two-phase interface boundary	May apply to simple cases involving phase change by interface reconstruction schemes.

This paper presents a CFD approach for analysis of water vapor condensation and two-phase flow between parallel plates. The model is developed by directly solving the governing equations for the entire single- and two-phase flow regions. Tracking of water phase change and two-phase flow is reached by employing a water liquid-phase fraction factor, defined as the ratio of the total enthalpy differential to the latent heat of condensation. The physical and transport properties of the two-phase region are calculated and updated based on the calculated value of the liquid-phase fraction factor until a converged result is reached. It is revealed that, among others, the inlet vapor velocity has significant effects on the water phase change and two-phase flow in the channel.

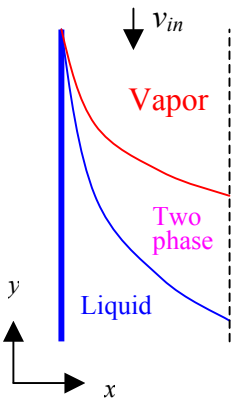

Figure 1: Physical model of water condensation in a channel of plate heat exchangers.

The water liquid-vapor phase interface due to the condensation of water vapor is investigated in this study for a geometric configuration of two parallel plates.

Because of its symmetry, only the left half of the channel is solved as shown in Fig. 1. The saturated vapor enters the channel with a plate length of L. Condensation takes place on the wall of the channel since the wall temperature, T_w, is below the saturation temperature of the water vapor, T_{sat}. The condensate water flows along the main flow direction due to the effects of gravity, shear force and surface tension. In a first attempt to implement the phase change and two-phase flow in a model, the following assumptions are employed during the modeling development: flow both in vapor and liquid phase is laminar; the fluid within the liquid-vapor interface is saturated; surface tension force is neglected (with including surface tension, the condensation length decreases, see [5]).

3 Governing equations and source terms with phase change

The conservation equations for transient, fluid flow in two dimensions are formulated in this section. In order to simplify the solution procedure, one set of governing equations is written for both the water liquid and vapor regions. The mass continuity equation or mass conservation equation is written as

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \mathbf{v}) = 0 \tag{1}$$

The momentum equations read

$$\frac{\partial (\rho u)}{\partial t} + \nabla \bullet (\rho u \mathbf{v}) = -\frac{\partial p}{\partial x} + \nabla \bullet (\mu \nabla u) \tag{2}$$

$$\frac{\partial (\rho v)}{\partial t} + \nabla \bullet (\rho v \mathbf{v}) = -\frac{\partial p}{\partial y} + \nabla \bullet (\mu \nabla v) - \rho g \tag{3}$$

where $\mathbf{v}$ is the velocity vector. The energy equation can be expressed as

$$\frac{\partial (\rho c_p T)}{\partial t} + \nabla \bullet (\rho c_p \mathbf{v} T) = \nabla \bullet (k \nabla T) - S_T \tag{4}$$

Equation (4) balances the convected energy, the heat conduction, and a source term S_T. The heat source term S_T in eqn (4) is associated with water phase change (condensation /vaporization), which can be expressed as [7]

$$S_T = \frac{\partial (\rho \Delta H)}{\partial t} + \nabla \bullet (\rho \Delta H \mathbf{v}) \tag{5}$$

where H is the total enthalpy. For a pure substance undergoing evaporation or condensation, the total enthalpy is a discontinuous function of the temperature. However, from a computational viewpoint, discontinuities are difficult to track and often the phase change is smeared out over a small temperature range to attain numerical stability [7]. The following relation is thus formulated for the total enthalpy H:

$$H = \frac{1}{\rho}\left[f \rho_v (h_v + h_{fg}) + (1-f)\rho_l h_l \right] = \frac{1}{\rho}\left[f \rho_v h_v + (1-f)\rho_l h_l \right] + \frac{1}{\rho} f \rho_v h_{fg} \tag{6a}$$

where f represents the liquid water fraction factor. The two terms on the right-hand side of eqn (6a) represent contributions of water sensible enthalpy, h, and latent heat, ΔH, respectively, to the total enthalpy

$$h = \frac{1}{\rho}\left[f\rho_v h_v + (1-f)\rho_l h_l \right] \tag{6b}$$

$$\Delta H = \frac{1}{\rho} f \rho_v h_{fg} \tag{6c}$$

The total enthalpy in eqn (6) can then be rewritten in terms of sensible enthalpy and latent heat

$$H = h + \Delta H \tag{7}$$

It should be pointed out that H is the enthalpy identical to that obtained from the steam table. It is equal to sensible enthalpy, h_l, for the liquid phase. For the vapor phase, H is the summation of sensible enthalpy h_v and latent heat of condensation h_{fg}. Substituting eqn (6c) into eqn (5) yields

$$S_T = \frac{\partial(\rho_v f h_{fg})}{\partial t} + \rho_v h_{fg} \nabla \bullet (f\mathbf{v}) \tag{8}$$

For incompressible flow, the water vapor density is a constant, and eqn (8) is simplified [5]

$$S_T = \rho_v h_{fg} \left[\frac{\partial f}{\partial t} + \nabla \bullet (f\mathbf{v}) \right] \tag{9}$$

The liquid water and vapor regions are distinguished by the liquid water fraction factor $f(x,y,t)$. The value of f is zero in the vapor phase and unity in the liquid phase. A formulation expressing the liquid phase fraction as a function of the total enthalpy, rather than the temperature, is employed in this study. According to the classical thermodynamic definition, the relationship between the liquid phase fraction f and the enthalpy H is

$$f = 1 - \frac{H - H_l}{H_v - H_l} \tag{10}$$

The time and coordinate dependence of the factor f follows the continuity law, and is expressed as

$$\frac{\partial f}{\partial t} + \nabla \bullet (f\mathbf{v}) = \frac{\dot{m}}{\rho_l} \tag{11}$$

where $\dot{m}$ is the mass production rate of condensate. It is clear that eqn (11) does not explicitly include the diffusive terms that smear the sharp gradient of the liquid water fraction factor f across the interface. The equation is added to the other governing equations with the purpose to predict the local and instantaneous value of f and other dependent parameters, and in the entire region smooth out the singularity encountered at the interface while preserving the phase discontinuity.

By comparing eqns (9) and (11), it is clear that the liquid water mass production rate due to condensation is associated with the source term in the energy equation according to

$$\dot{m} = \frac{\rho_l}{h_{fg}\rho_v} S_T \tag{12}$$

4 Fluid properties

The fluid properties at the interface vary with time as the interface position changes relative to the computational grid. The density, viscosity and thermal conductivity are defined as functions of f according to the following relations [6]:

$$\rho = f\rho_l + (1-f)\rho_v \tag{13}$$

$$\mu = f\mu_l + (1-f)\mu_v \tag{14}$$

$$k = fk_l + (1-f)k_v \tag{15}$$

The specific heat is defined using the weight fraction of water liquid and vapor

$$c_p = \frac{1}{\rho}\left[f\rho_l c_{pl} + (1-f)\rho_v c_{pv} \right] \tag{16}$$

5 Boundary conditions

The inlet conditions can be written as

$$u=0,\ v=-v_{in},\ T=T_{in}=T_{sat},\ f=0 \text{ at the top of the channel } (y=L) \tag{17}$$

while the boundary conditions at the wall (x=0) are

$$u=v=0,\ T=T_w,\ f=1 \text{ at } x=0 \tag{18}$$

The boundary conditions at the exit of the channel is written in a generalized form, i.e., $\partial\phi/\partial y = 0$ at y=0 for u, v, T or f. Due to the symmetric character, only one half of the channel is considered. Along the line of symmetry ($x = W$) the boundary conditions can be expressed as $\partial\phi/\partial x = 0$.

6 Numerical solution methodology

The governing differential equations are discretized into algebraic equations by a truncated Taylor series approach using the finite volume method (see, e.g., [10]), and then they are solved by an iterative method. A general two-dimensional CFD code, SIMPLE_HT [11], is employed to solve the governing eqns (1–4, 11), together with the boundary conditions. The finite-volume (or control-volume) technique, which is based on the conservation of a specific physical quantity, is used in the code. The code is designed for convection-diffusion problems, e.g., Navier-Stokes equations, the temperature field and other variable equations. As implemented in the code, each of the differential equations can be cast into the general equation form (see, e.g., [11])

$$\frac{\partial}{\partial \tau}(\rho\phi)+\frac{\partial}{\partial x}(\rho u\phi)+\frac{\partial}{\partial y}(\rho v\phi)=\frac{\partial}{\partial x}\left(\Gamma\frac{\partial \phi}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma\frac{\partial \phi}{\partial y}\right)+S \qquad (19)$$

where ϕ denotes any of the dependent variables, Γ is the diffusivity and S is a source term. Once in this form, the equations are integrated over the control volumes defined on a staggered grid. The boundary conditions are introduced as source terms in control volumes neighboring boundaries whenever appropriate. The resulting system of algebraic equations is solved using an iterative method. The pressure-velocity coupling (when $\phi = u, v$) is treated by the SIMPLEC-procedure with the incompressible form of the pressure-correction equation. The convection-diffusion terms are treated by the power-law, hybrid or upwind (used in this study) schemes. Each variable is solved with the TDMA algorithm combined with a block-correction method.

As shown above, the equations needed for the calculation are coupled by the liquid phase fraction factor *f*, temperature, condensation mass generation via source terms and variable physical/transport properties. It should be noted that the mass generation rate is zero in most of the region, and non-zero only in the region of two-phase flow, where phase change occurs. As mentioned earlier, the physical/transport properties are variable. These parameters depend on the position in the channel, and the liquid phase fraction factor as well. All the parameters are calculated and updated during iterations of the calculation procedure. In this investigation, a uniform grid distribution is applied.

In order to evaluate the performance of the numerical method and code, test calculations considering grid sensitivity were carried out. It is found that the predictions do not change significantly in terms of the liquid phase fraction factor distribution, when the number of grid points is increased beyond 18×70 (70 for the main flow direction *y*).

It should be mentioned that water liquid and vapor temperatures are continuous at the interface. The temperature at the liquid-vapor interface is therefore expressed as $T = T_{sat}$, where T_{sat} is the saturation temperature. If a fixed value is desired for a variable anywhere in the computational domain, this can be achieved by introducing a fictious source term for the cells with *f* between 0 and 1.

$$S_T = 10^{10} T_{sat} - 10^{10} T \qquad (20)$$

By doing this, the source term will dominate and the interface temperature T is set to the saturation temperature T_{sat}.

7 Results and discussion

Parameters of a PHE configuration, appearing in the common literature, are applied as a base case in this study. Channel geometries are employed as follows: length of the channel L = 20 cm, and width of the channel $2W$ = 4 mm. The inlet vapor is assumed to be saturated, and the inlet conditions are: temperature T_{in} = 100°C and velocity v_{in} = 0.6 m/s. It should be noted that all the

results presented hereafter are for the base case condition unless otherwise stated, such as the inlet velocity.

The velocity vector and liquid phase fraction factor f distributions are shown in Fig. 2. Figure 2(a) shows the velocity profile along the main flow direction, in which the scale of the vector plots (i.e., 0.5 m/s) is a reference value of the maximum velocity. As shown in the figure, a parabolic profile is observed in the flow channel. On the other hand, the velocity in the liquid region close to the plate is very small. In terms of liquid water fraction factor f, water vapor condensation and two-phase flow phenomenon in the channel are shown in Fig. 2(b). It can be observed that water vapor with $f = 0$ appears in the central part of the channel along the main flow direction, while generated liquid water ($f = 1$) due to condensation is found in the region closest to the plate. The liquid-vapor interface with $0<f<1$ is between these two regions. It is clear that the thickness of both the liquid-vapor interface and condensate liquid water layer is becoming larger along the main flow direction.

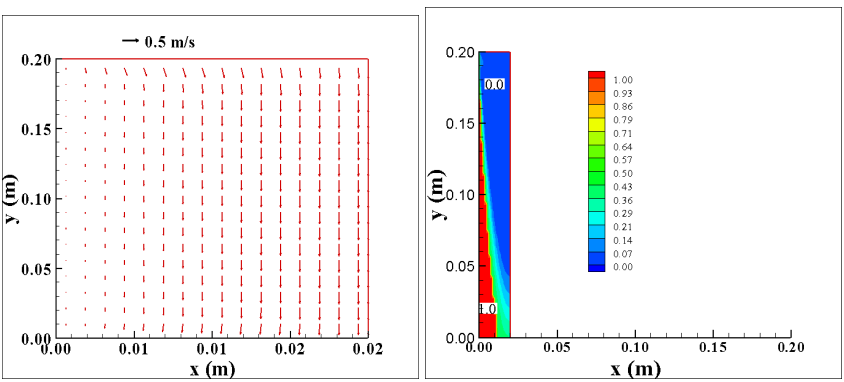

Figure 2: (a) Velocity vector, and (b) contour of liquid phase fraction factor for the base conditions.

Fig. 3 shows the distribution of water liquid phase fraction factor f at different vapor inlet velocities. For a small vapor inlet velocity ($v_{in} = 0.3$ m/s) as shown in Fig. 3(a), all the water vapor supplied to the channel condenses, and the generated liquid water fully occupies the flow region at the exit of the channel. On the other hand, for big vapor inlet velocity as shown in Fig. 3(b), both the condensate liquid water layer and the two-phase interface are very thin, and most of the supplied water vapor flows out the channel exit. For this case, a modest phase change (condensation) occurs and weak effects on the two-phase flow are expected. It is obvious that a big downward vapor velocity will tend to thin the liquid water film, and this is the reason why most condensation applications occur in a downward style [1].

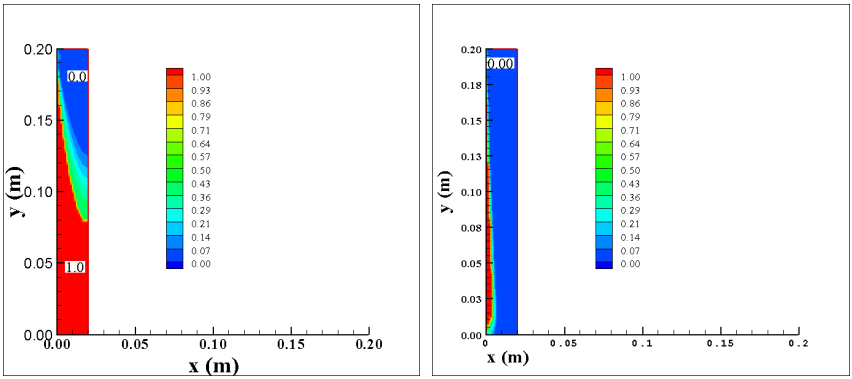

Figure 3: Contours of liquid phase fraction factor for the vapor inlet velocity v_{in} of: (a) 0.3 m/s, and (b) 1.2 m/s.

8 Conclusion

In this investigation, water condensation and two-phase flow interface are numerically simulated for a channel between two parallel plates relevant for PHEs. The model is based on the total enthalpy of water, and water liquid phase fraction factor is employed to describe the water vapor, liquid and two-phase interface. The results show that the condensation phase change and two-phase interface are very sensitive to the vapor inlet velocity. It should be pointed out that the surface tension is not included in this study, and its effects on the distribution of liquid water fraction factor will be evaluated later.

Acknowledgements

The Swedish Energy Agency (STEM) and the Swedish Research Council (VR)/SIDA -Swedish Research Links provided financial support.

References

[1] Wang, L. and Sundén, B., Thermal and Hydraulic Performance of Plate Heat Exchangers as Condensers, in Proceedings: *Compact Heat Exchangers and Enhancement Technology for the Process Industries*, Shah, R.K. (ed.), pp 461-469, 2003.

[2] Panday, P.K., Two-dimensional Turbulent Film Condensation of Vapours Flowing inside a Vertical Tube and between Parallel Plates: a Numerical Approach, *Int. J. Refrigeration*, **26**, pp 492-503, 2003.

[3] Srzic, V., Soliman, H.M. and Ormiston, S.J., Analysis of Laminar Mixed-convection Condensation on Isothermal Plates Using the Full Boundary-layer Equations: Mixtures of a Vapor and a Lighter Gas, *Int. J. Heat Mass Transfer,* **42**, pp. 685-695, 1999.

[4] White, A.J., Numerical Investigation of Condensing Steam Flow in Boundary Layers, *Int. J. Heat Fluid Flow,* **21**, pp. 727-734, 2000.
[5] Zhang, Y., Faghri, A. & Shafii, M.B., Capillary Blocking in Forced Convective Condensation in Horizontal Miniature Channels, ASME Transaction *J. Heat Transfer*, **123**, pp. 501-511, 2001.
[6] Zhang, Y. & Faghri, A., Numerical Simulation of Condensation on a Capillary Grooved Structure, *Num. Heat Transfer* (part A), **39**, pp. 227-243, 2001.
[7] Anghaie, A., Chen, G. & Kim, S., An Energy Based Pressure Correction Method for Diabatic Two-Phase Flow with Phase Change, in Proceedings: *Trends in Numerical and Physical Modeling for Industrial Multiphase Flows*, Institut d'Etudes Scientifigues de Cargese, France, Sept. 27-29, 2000.
[8] Wang, L. & Sundén, B., Numerical Simulation of Two-phase Fluid Flow and Heat Transfer With or Without Phase Change Using a Volume-of Fluid Model, in Proceedings: *2004 ASME Int. Mechanical Engineering Congress and Exposition* (CD-rom), IMECE2004-59380, UAS, 2004.
[9] Wang, L. & Sundén, B., Numerical Simulation of Two-phase Flows Using a Volume-of-Fluid Model with Various Interface Reconstruction Schemes, in Proceedings: *3rd Int. Symposium on Two-Phase Flow Modeling and Experimentation*, Pisa, 22-24 Sept. 2004.
[10] Versteeg, H.K., and Malalasekera, W.M., *An Introduction to Computational Fluid Dynamics, the Finite Volume Method*, Longman Scientific & Technical, England, 1995.
[11] Sundén, B., Rokni, M., Faghri, M. and Eriksson, D., *The Computer Code SIMPLE_HT*, ISRN LUTMDN/TMHP-04/3015-SE, Lund Institute of Technology, Lund, Oct. 2004.