International Journal of Heat and Mass Transfer 85 (2015) 865–879

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Experimental and computational investigation of vertical downflow condensation

Hyoungsoon Lee, Chirag R. Kharangate, Nikhin Mascarenhas, Ilchung Park, Issam Mudawar *

Purdue University Boiling and Two-Phase Flow Laboratory (PU-BTPFL), Mechanical Engineering Building, 585 Purdue Mall, West Lafayette, IN 47907-2088, USA

ARTICLE INFO

Article history:
Received 1 July 2014
Received in revised form 10 February 2015
Accepted 12 February 2015

Keywords:
Condensation
Computational two-phase flow
Annular flow
Vertical downflow

ABSTRACT

This explores downflow condensation in a circular tube both experimentally and computationally using FC-72 as a working fluid. A highly instrumented condensation module is used to map detailed axial variations of both wall heat flux and wall temperature, which are used to determine axial variations of the condensation heat transfer coefficient. The experimental results are compared to predictions of a two-dimensional axisymmetric computational model using FLUENT. The study provides detailed construction of the model, including choice of interfacial phase change sub-model, numerical methods, and convergence criteria. The model is shown to yield good prediction of the heat transfer coefficient. The computed temperature profiles exhibit unusual shape, with steep gradient near the annular liquid film interface as well as near the wall, and a mild gradient in between. This shape is shown to be closely related to the shape of the eddy diffusivity profile. These findings point to the need for future, more sophisticated measurements of liquid film thickness, and both velocity and temperature profiles, to both validate and refine two-phase computational models.

1. Introduction

1.1. Importance of predictive tools for two-phase cooling system design

Two-phase cooling systems have gained significant popularity in recent years due to an urgent need to tackle unprecedented heat dissipation challenges in commercial, aerospace and defense electronics. Heat dissipation rates at the device and system levels in these applications are no longer manageable with single-phase cooling systems. Therefore, more aggressive two-phase cooling solutions are being sought, using a variety of configurations, such as pool [1–3], mini/micro-channel [4–7], jet [8–10] and spray [11,12], as well as hybrid configurations combining the benefits of two or more of these configurations [13,14]. These two-phase cooling solutions have already yielded as high as 1127 W/cm^2 using dielectric coolants [15] and 27,600 W/cm^2 using water [16,17].

A complete two-phase cooling system requires an effective means for rejecting the heat absorbed from the electronics to the ambient, and this is generally achieved with the aid of a high performance condenser. Unfortunately, recent studies concerning the development of two-phase cooling solutions have focused mostly on the heat acquisition by boiling, and to a far lesser degree on ultimate rejection of the heat by condensation. Clearly, a better understanding of the thermal performance of high performance condensers is needed to achieve a more complete methodology for cooling system design. Recent findings concerning fluid flow and heat transfer mechanisms of condensation are available in Refs. [18–20].

While empirical formulations constitute the most popular means for predicting transport behavior in condensers [21–23], these formulations are limited by working fluids, geometries, and operating conditions of the databases upon which they are based. To achieve more universal predictive tools, there is a desire to develop both theoretical and computational techniques. Theoretical models are based mostly on the application of mass, momentum and energy conservation laws to control volumes encompassing the liquid and vapor phases or the entire flow [24]. These models have been especially effective for annular condensation, but far less so for slug flow and bubbly flow. But even for annular flow, there are major modeling challenges stemming from a lack of ability to model interfacial waves and the influence of the interface on turbulence in both the liquid film and vapor core [24–26].

* Corresponding author. Tel.: +1 (765) 494 5705; fax: +1 (765) 494 0539.
E-mail address: mudawar@ecn.purdue.edu (I. Mudawar).
URL: https://engineering.purdue.edu/BTPFL (I. Mudawar).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.02.037
0017-9310/

Nomenclature

c_p	specific heat at constant pressure
c	mesh (cell) size
D	diameter
E	energy per unit mass
F	force
G	mass velocity
g	gravitational acceleration
H	step function
h	heat transfer coefficient
h_{fg}	latent heat of vaporization
I	turbulence intensity
j_e^h	heat flux from wall to interface in Schrage model
k	thermal conductivity
L	length
M	molecular weight
$\dot{m}''$	mass flux
n	number of cells
$\vec{n}$	unit vector normal to interface
P	pressure
Pr	Prandtl number
Pr_T	turbulent Prandtl number
Q	energy source term
q''	heat flux
R	universal gas constant (8.314 J/mol K)
r	radial coordinate
Re	Reynolds number
r_i	mass transfer intensity factor
S	volumetric mass source
T	temperature
t	time
t^+	dimensionless time
U	mean inlet vapor velocity
u	velocity
u'	fluctuating component of velocity
u^*	friction velocity $\sqrt{\tau_{wall}/\rho_f}$
V	volume
x_e	thermodynamic equilibrium quality
y	distance perpendicular to wall
y^+	dimensionless distance perpendicular to wall
z	stream-wise coordinate
z_0	location where $x_e = 1$

Greek symbols

α	volume fraction, void fraction
γ	accommodation coefficient in Schrage model
δ	liquid film thickness
ε_m	eddy momentum diffusivity
η_e	evaporation heat transfer coefficient in Schrage model
μ	dynamic viscosity
ν	kinematic viscosity
ρ	density
σ	surface tension
τ	shear stress
φ	level set function
ψ	generic property

Superscript

–	average component

Subscripts

c	condensation
D	diameter
e	evaporation; thermodynamic equilibrium
eff	effective
exp	experimental
f	liquid
g	vapor
i	initial; interfacial
in	inlet
sat	saturated
w	water
$wall$	wall

1.2. Application of commercial software and numerical methods to two-phase systems

The limitations of empirical formulations and theoretical models point to the need for computational tools that can tackle interfacial waviness, turbulence structure, and different condensation flow patterns. Computational tools have been quite effective in simulating complex turbulent single-phase systems, showing good agreement with experimental data. And computation efforts have benefitted greatly from both the developments in CPU processing speed and availability of commercial CFD software such as ANSYS FLUENT, OpenFOAM, FLOW-3D and COMSOL. Increasingly, these software packages have been able to combine various numerical techniques to enhance prediction accuracy. Aside from the ability to tackle complicated geometries, computational tools can significantly reduce cost, especially for efforts where extensive testing is cost prohibitive such as microgravity experiments [27]. However, the ability of commercial software to accurately predict complex two-phase flows is questionable, which is why many researchers are making persistent efforts in pursuit of improved accuracy.

Numerical approaches to modeling two-phase systems have been studies quite thoroughly and several types of approaches have been developed for this purpose. These approaches can be categorized into three different types: (1) Lagrangian, (2) Eulerian and (3) Eulerian–Lagrangian.

Smoothed-Particle Hydrodynamics (SPH) [28,29] and Multiphase Particle-in-Cell (MP-PIC) [30] methods are based on the Lagrangian approach. These methods provide a high level of accuracy at the interface and allow interfacial boundary conditions to be easily applied because they can tackle individual fluid particles and determine how these particles behaves in motion. However, Lagrangian methods are limited to simple cases due to the complexity of applying complicated grid topologies. Therefore, as the demand for solving complex situations increases, methods following Eulerian perspectives have gained more popularity, given their simplicity, feasibility of tackling multiple interfaces (e.g., multiple bubbles), and relative ease of implementation into commercial CFD software.

Level-Set Method (LSM) [31] and Volume-of-Fluid (VOF) method [32] are two representative examples based on the Eulerian perspective. Developed by Osher and Sethian [31], LSM tracks the interface by a smooth function φ, where $\varphi = 0$ at the interface, which is called the zero level set, and is positive for one phase and negative for the other. In this method, interfacial topologies such as curvatures and sharp interfaces can be easily captured. However, this method suffers an inability to tackle mass conservation, as it results in loss of mass when solving the