International Journal of Heat and Mass Transfer 126 (2018) 1194–1205

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Modeling of film condensation flow in oval microchannels

Ji Wang [a], Jun Ming Li [a,*], Yunho Hwang [b]

[a] Key Laboratory for Thermal Science and Power Engineering of Ministry of Education, Department of Energy and Power Engineering, Tsinghua University, Beijing 100084, China
[b] Center for Energy Environmental Engineering, Department of Mechanical Engineering, University of Maryland, College Park, MD 20740, United States

ARTICLE INFO

Article history:
Received 11 December 2017
Received in revised form 18 May 2018
Accepted 24 May 2018

Keywords:
Condensation
Microchannel
Theoretical modeling
Refrigerant
Heat transfer

ABSTRACT

A theoretical film condensation heat transfer model was developed for refrigerant flowing through oval-shaped microchannels in annular flow regime with laminar liquid film flowing along the channel walls. The model considered the effects of surface tension, disjoining pressure, interfacial shear stress and interfacial thermal resistance. Study was conducted for mass fluxes in a range of 300–1500 kg/(m^2 s) with two refrigerants, R134a and R1234ze(E). The local heat transfer coefficient and circumferential mean heat transfer coefficient along the flow direction were obtained. The liquid film thickness profiles at different locations are presented. Due to the effect of surface tension, the liquid film accumulates towards the semi-circular corner. The circumferential mean heat transfer coefficient is very high after entering the channel inlet, decreases along the flow direction and reaches a constant value. R134a has a slightly higher circumferential mean heat transfer coefficient than that of R1234ze(E) due to its higher liquid conductivity.

1. Introduction

Recently microchannel condensers have been used in many applications such as air-conditioning, refrigeration system and chemical processes due their high heat transfer performance in compact design. Design of microchannel heat condensers needs an accurate heat transfer model. However, the temperature inside the microchannels could not be directly measured due to their relatively small dimensions. Therefore, in the early experimental studies on condensation heat transfer in minichannels and microchannels, heat transfer coefficient was obtained by measuring overall refrigerant and coolant temperatures, which has a large uncertainty. Researchers, such as Yang and Webb [1], Kim et al. [2], Yan and Lin [3], Wang et al. [4] and Garimella and Bandhauer [5], have used this method. Recently, some researchers attempted to study the condensation heat transfer in minichannels and microchannels by measuring the channel outer surface temperature. Using this method, Koyama et al. [6] and Cavallini et al. [7] measured the heat transfer coefficient in multiport channels, while Matkovic et al. [8], Del Col et al. [9], Cavallini et al. [10], Zhang et al. [11] and Liu et al. [12–14] measured the heat transfer coefficient in single channels. These researchers all used refrigerants and channels with hydraulic diameters of around 1 mm. Available empirical and semi-empirical correlations developed for conventional channels (with hydraulic diameter larger than 1 mm) could not predict their data successfully, therefore some researchers modified the correlations based on their own data. With the development of silicon etching technology, people could manufacture smaller channels on silicon wafers, which makes it possible to study condensation heat transfer in microchannels with hydraulic diameters much smaller than 1 mm. By measuring the wall temperature of the silicon microchannels, steam condensation heat transfer coefficient was obtained by Wu et al. [15], Chen et al. [16], Wu et al. [17], Chen et al. [18] and Quan et al. [19] in microchannels with several cross-sectional shapes and hydraulic diameters smaller than 300 μm. However, due to its low-pressure tolerance, silicon microchannels could not be used to study condensation heat transfer of refrigerants with high saturation pressure. Only Dong and Yang [20] studied with refrigerant R141b.

In addition, researchers also modeled condensation flow in channels by assuming laminar flow for liquid film. Begg et al. [21], Wang and Du [22] and Miscevic et al. [23] modeled annular condensation flow in circular channels. The heat transfer in non-circular channels is better than that in circular channels with the same hydraulic diameter. The enhancement of condensation heat transfer in non-circular microchannels stems from the transverse pressure gradient in the liquid film caused by surface tension. As a result, liquid tends to flow towards channel corners, which thins liquid film at the flat channel surface and reduces the thermal resistance. This mechanism was analyzed theoretically by several researchers as follows. Zhao and Liao [24] calculated film conden-

* Corresponding author.
E-mail address: lijm@tsinghua.edu.cn (J.M. Li).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.05.126
0017-9310/

Nomenclature

a	cross-sectional dimension of the channel, which is 0.15 mm
A_v	vapor flow area (m^2)
Cp	heat capacity at constant pressure (J/(kg·K))
Cv	heat capacity at constant volume (J/(kg·K))
D	hydraulic diameter (m)
f	friction factor
G	refrigerant mass flux (kg/(m^2 s))
h	heat transfer coefficient (W/(m^2 K))
L	latent heat (J/kg)
H	Hamaker parameter (J)
m	mass flux at the liquid-vapor interface due to condensation (kg/(m^2 s))
$\hat{n}$	unit normal of surface
O	origin of the cylindrical coordinate
p	pressure (Pa)
r	radius, radial polar coordinate (m)
Re	Reynolds number
R_g	ideal gas constant (J/(kg·K))
S_v	perimeter of the vapor-liquid interface in channel cross section (m)
T	temperature (°C)
u	liquid velocity along channel surface in the x direction or φ direction (m/s)
U	average velocity (m/s)
v	liquid velocity along channel surface in the z direction (m/s)
x	coordinate along the channel surface (m)
y	coordinate normal to the channel surface (m)
z	coordinate along the flow direction (m)

Greek symbols

ε	surface roughness (m)
γ	heat capacity ratio
κ	thermal conductivity (W/(m·K))
μ	dynamic viscosity (Pa·s)
ν	kinematic viscosity (m^2/s)
ξ	constant used to calculate the interfacial thermal resistance
ρ	density (kg/m^3)
σ	surface tension (N/m)
δ	liquid film thickness (m)
ς	liquid-vapor interfacial thermal resistance (m^2 K/W)
τ	shear stress (Pa)
φ	angle, polar coordinate (rad)
ϕ	see Eq. (12)
χ	vapor mass quality

Subscripts

0	position of $x = 0$
c	curvature
i	liquid-vapor interface
in	channel inlet
l	liquid
m	position of $\varphi = \pi/2$
s	saturation
v	vapor
w	channel wall

sation in a vertical triangular channel by dividing the channel cross section into liquid film, liquid meniscus and vapor core zone. Similar models were developed by Wu et al. [25] for rectangular channels and Hao et al. [26] for trapezoidal, rectangular and triangular channels. Wang and Rose [27,28] systematically calculated condensation flow in horizontal non-circular channels with the presence of gravity. They modeled thin liquid film zone and meniscus zone using Cartesian coordinate and cylindrical coordinate, respectively. They obtained liquid film thickness in rectangular, triangular and circular channels, and took into account the gravitational effects for channels with different orientations [29]. Based on their calculation results, they proposed a heat transfer coefficient correlation for condensation flow in horizontal non-circular channels [30]. Nebuloni and Thome [31] also calculated condensation flow in horizontal non-circular channels with gravitational effects. They used finite volume difference method to solve Navier-Stokes and energy equations in liquid film and meniscus zone. They obtained liquid film thickness in circular, elliptic and flattened channels and analyzed the effect of non-uniform heat flux [32].

With the advancement of Computational Fluid Dynamics (CFD) technology, Volume of Fluid (VOF) model was also used to simulate in-tube condensation flow, such as the studies presented by Zhang et al. [33], Da Riva et al. [34,35], Chen et al. [36] and Ganapythy et al. [37]. The problem of VOF model for condensation flow simulation is the high computational cost due to the need for a fine grid in the thin liquid film.

As summarized above, none of the previous studies has investigated for condensation flow in oval-shaped microchannel. Therefore, this paper presents a theoretical model of annular flow in a specific oval-shaped microchannel, which can be manufactured using isotropic wet itching [38]. The channel dimension is so small that the condensation heat transfer coefficient measurement is very difficult. Therefore, numerical modelling is used to analyze the heat transfer characteristics in it. The gravitational effect was neglected, while the surface tension, disjoining pressure, interfacial shear stress and interfacial thermal resistance were taken into account. The model was developed based on the study of Wang and Rose [27,28]. Different from the previous research, the curvature expression for surface tension calculation was based on a 3-dimensional surface, and the inlet of the channel was assumed to be well-developed adiabatic flow. The effect of mass flux on local heat transfer coefficient in the microchannel was analyzed for R134a and R1234ze(E).

2. Theoretical model

The geometry and coordinates of the microchannel are illustrated in Fig. 1. Fig. 1(a) shows cross-sectional views of the microchannel, which is given by a scanning electron microscope. As the gravitational effect is neglected in the current model, the liquid film distribution in the channel is symmetric. Therefore, to simplify the model, a quarter of the channel cross section is considered in the physical model, as shown in Fig. 1(b). Cartesian coordinate (x, y) and cylindrical coordinate (φ, r) are used for the flat sidewall ($0 < x < a$) and semi-circular corner, respectively ($0 < \varphi < \pi/2$). $a = 0.15$ mm and $r_w = 0.1$ mm are corresponding to the dimensions in Fig. 1(a). In the model, the following assumptions are made:

- The liquid flow along x- (or φ-) and z-direction is laminar flow;
- The heat transfer through the liquid film is only conduction, and the convection is negligible.
- The vapor flow is a one-dimensional fully developed flow along z-direction.