International Journal of Heat and Mass Transfer 79 (2014) 561–572

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Three-dimensional simulation on behavior of water film flow with and without shear stress on water–air interface

Y.Q. Yu [a,c,*], X. Cheng [a,b]

[a] School of Nuclear Science and Engineering, Shanghai Jiao Tong University, Shanghai 200240, China
[b] Institute for Nuclear and Energy Technologies Research Center Karlsruhe, Karlsruhe 76021, Germany
[c] Nuclear Engineering Division, Argonne National Laboratory, Lemont, IL 60439, United States

ARTICLE INFO

Article history:
Received 13 January 2014
Received in revised form 11 August 2014
Accepted 13 August 2014
Available online 7 September 2014

Keywords:
Falling film
Solitary wave
Flat plate
MWR
MLFT

ABSTRACT

In this paper, the simulations of falling film behavior on a flat plate with and without interfacial gas–liquid shear stress were carried out. A three-dimensional numerical model was established based on film flow characteristics. A source term was implanted into the numerical model to take into account the interfacial gas–liquid shear stress. The model was validated by the experimental data. Both continuous film flow and film breakup were simulated. The film thickness, velocity distribution and wall shear stress at different Reynolds numbers were presented to understand the film flow behavior comprehensively. The influence of water–air shear stress on film flow behavior was revealed. A reasonable prediction on both MWR (Minimum Wetting Rate: Δ_{min}) and MLFT (Minimum Liquid Film Thickness: Γ_{min}) were obtained. The model proposed in this study ought to be profitable for studies on mechanism of film breakup.

1. Introduction

Film flow is widely applied in many industrial fields due to its high thermal efficiency and low flow rate. The application in Passive Containment Cooling System (PCCS) of Generation III nuclear power plant AP1000 is one of them. Driven by gravity in PCCS, the water from a storage tank is poured on the containment and forms a film over it during the accidents. The film flow on the containment plays a significant role on decay heat removal. In the meantime, the nature convection in PCCS can cause gas–liquid shear stress on the free surface of the film flow. Due to large diameter of the containment, the film flow on the containment can always be considered as film flow on a flat plate.

Film flow has been extensively studied for several decades. Although current measuring techniques have been improved considerably, the experimental studies on film flow are still confined in measurement of wave velocity, wave length, and film thickness. The flow field inside film flow is difficult to measure due to the tiny scale of film thickness. Consequently, numerical study becomes a promising way to provide better understand on flow behavior inside the film. Nusselt [1] proposed a laminar model for film flow. The model assumes that film flow is laminar with smooth surface and the shear stress on the interface is 0. The Navier–Stokes equation for film flow can be expressed as follows:

$$\rho\frac{DU}{Dt} = -\nabla P + \rho g + \mu\nabla^2 U \quad (1)$$

According to the assumption of Nusselt theory:

$$U_x = U_y = 0 \quad (2)$$

$$\frac{\partial U_z}{\partial z} = 0 \quad (3)$$

where z is the direction of film flow, x is the width direction and y is the direction of film thickness, U is the velocity of film flow, μ is the molecular viscosity.

The Eq. (1) can be simplified with Eqs. (2) and (3):

$$\mu\frac{d^2U_z}{dy^2} = -\rho g \quad (4)$$

No slip wall and no interfacial sheer stress boundary condition can be expressed as follows:

$$y = 0, \quad U_z = 0 \quad (5)$$

$$y = \delta, \quad \frac{dU_z}{dy} = 0 \quad (6)$$

where δ is film thickness. A parabolic velocity distribution inside film flow can be obtained by substituting Eqs. (5) and (6) for Eq. (4):

* Corresponding author at: School of Nuclear Science and Engineering, Shanghai Jiao Tong University, Shanghai 200240, China. Tel.: +1 630 252 1604.
E-mail address: yyu@anl.gov (Y.Q. Yu).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.08.035
0017-9310/

$$U_z(y) = \frac{\rho g \delta^2}{2\mu}\left[\frac{2y}{\delta} - \left(\frac{y}{\delta}\right)^2\right] \tag{7}$$

Based on the principle of mass conservation, velocity distribution and definition of Reynolds number of film flow (Eq. (8)), the relationship between film thickness and Reynolds number can be obtained in Eq. (9).

$$\mathrm{Re} = \frac{4\Gamma}{\mu} \tag{8}$$

$$\delta = \left(\frac{3}{4}\right)^{1/3} Re^{1/3}\left(\frac{\nu^2}{g}\right)^{1/3} \tag{9}$$

where Γ is the linear mass velocity (kg/m s), Re is Reynolds number and υ is the dynamic viscosity of film flow (m^2/s).

The instability is one of the most significant characteristic of film flow, which leads to the production of surface waves. When film is falling down, it is influenced by small disturbances which will gradually develop to large surface waves. Generally speaking, film flow consists of film substrate and surface wave which is also referred to as solitary wave. The fluid in the film substrate is laminar while the fluid can be turbulent in the solitary wave. Most theoretical studies are based on smooth surface assumption, the fidelity of which is poor. In recent years, Computational Fluid Dynamics (CFD) method has achieve some success on modeling a wide range of industrial applications, such as water–vapor flows in PWR/BWR, gas–solid fluidization, gasification, etc. [35,36,38,37]. Among the numerical studies, CFD is of higher fidelity to describe the film flow behavior in spite of its considerable computational cost. Fortunately, it becomes realizable with the development of computers in recent decades.

Kheshgi and Scriven [2] solved two dimensional Navier–Stokes equations to analyze the stability of film flow on a flat plate. The results showed that the disturbances fade away when wave length of the disturbance is smaller than some critical wave length. Otherwise, the disturbances will develop to steady waves. The large disturbances may probably lead to the decomposition of the wave and the film breakup, which agree well with other linear numerical results through solving Orr–Sommerfeld equation. CFD study [3] also indicated that the velocity distribution in capillary wave is similar with that in solitary wave. The vortices inside the solitary wave are formed when the surface velocity of the wave crest exceeds the wave velocity with the increase of the wave amplitude. It is also pointed out that the forming of the vortices requires the ratio between solitary wave height and substrate thickness to be over 2.5. Stuhlträger et al. [4] studied the effect of surface wave on heat transfer with CFD method. The numerical results revealed that the heat transfer coefficient is enhanced due to the decrease of the average film thickness. The time-averaged temperature presented linear distribution in film thickness direction. Serifi et al. [5] simulated the film flow on an inclined flat plate. The thermal hydraulic characteristics of film flow were analyzed. The research pointed out two main factors which enhance the heat transfer of film flow. One is the decrease of substrate thickness and the other is convection effect in wave crest and tail. Tomoaki and Chiaki [6] CFD calculation found out that the heat transfer of film flow can be enhanced by introducing artificial disturbance. Miyara [7–9] performed the CFD simulation on solitary wave with low Reynolds number. The research found out that the recirculation zone in the solitary wave intensifies the heat transfer between film and wall. The research also indicated that scale of the vortices increase with Reynolds number. Jayanti and Hewitt [10,11] studied the characteristics of solitary waves with CFD tool. The waves of three typical shapes were investigated, which is sinusoidal wave, twisted sinusoidal wave and solitary wave respectively. The results indicated that there is no recirculation zone under the wave for sinusoidal wave with small amplitude. As for sinusoidal wave with larger amplitude, the heat transfer coefficient is enhanced due to the thinner effective film thickness. The research also pointed out that the fluid in the substrate keeps laminar while the fluid in solitary wave is turbulent with strong turbulent diffusion. Gao et al. [12] used VOF (Volume Of Fluid)–CSF (Continuum Surface Force) method to simulate the free falling film flow under single frequency disturbance. The results confirmed the existence of vortices inside the solitary wave. Sutalo et al. [13] used ANSYS-CFX to simulate the non-Newtonian film flowing along an inclined plate. The calculated film thickness and film shape agree well with the experimental data.

CFD method requires solving huge amounts of nonlinear equations. Therefore, most CFD studies of film flow are two-dimensional. Only modest three-dimensional simulations on film flow were developed. Yoshida et al. [14] predicted both the average film and the evolution of the solitary waves well with three dimensional CFD model. The calculated probability density function (PDF) of film thickness agrees well with the experimental data. Haeria and Hashemabadi [15] studied the effect of film property, inclination and contact angle on film flow with open source CFD tool OpenFOAM. The velocity distribution and shape of film flow are obtained and in good agreement with the measured data. Lakehal et al. [16] applied DNS method to study the thermal hydraulic characteristics of film flow with counter current gas flow. The study indicated that the heat and mass transfer are tightly associated with the surface wave and the liquid–gas shear stress.

From the literature survey mentioned above, most current CFD study is two-dimensional. Many experimental researches have pointed out that film flow shows strong three-dimensional characteristics. Two-dimensional CFD simulations are not capable of predicting these characteristics. Thus, three-dimensional simulations on film flow have become a very meaningful research focus. However, among the existed rare three-dimensional simulations, most of them include no validation work or inadequate validation work. In order to achieve a better understanding of flow mechanisms on film flow, which is helpful for evaluating and improving the PCCS as well as other film flow applications, this paper describes a three-dimensional CFD study of falling film flow with and without shear stress on water–air interface. An adequate number of verification and validation work are performed for establishing the numerical model.

2. Numerical strategy

2.1. Governing equation

Water is incompressible Newtonian fluid. The velocity field of film flow on flat plate include three velocity component: $V = (u, \upsilon, w)$. The three-dimensional continuity equation for film flow on flat plate is:

$$\frac{\partial\bar\rho}{\partial t} + \frac{\partial\bar\rho u}{\partial x} + \frac{\partial\bar\rho \upsilon}{\partial y} + \frac{\partial\bar\rho w}{\partial z} = 0 \tag{10}$$

It is worth mentioning that this study applies the homogeneous model. Hence, air and water in whole computational domain are considered as one fluid. There is no velocity difference between two phases. The three-dimensional Navier–Stokes equations for film flow on flat plate are:

$$\frac{\partial\bar\rho u}{\partial t} + \frac{\partial\bar\rho uu}{\partial x} + \frac{\partial\bar\rho u\upsilon}{\partial y} + \frac{\partial\bar\rho uw}{\partial z} = -\frac{\partial p}{\partial x} + \bar\mu\nabla^2 u + F_{\sigma x} \tag{11}$$

$$\frac{\partial\bar\rho \upsilon}{\partial t} + \frac{\partial\bar\rho \upsilon u}{\partial x} + \frac{\partial\bar\rho \upsilon\upsilon}{\partial y} + \frac{\partial\bar\rho \upsilon w}{\partial z} = -\frac{\partial p}{\partial y} + \bar\mu\nabla^2 \upsilon - \bar\rho g cos\emptyset + F_{\sigma y} \tag{12}$$

$$\frac{\partial\bar\rho w}{\partial t} + \frac{\partial\bar\rho wu}{\partial x} + \frac{\partial\bar\rho w\upsilon}{\partial y} + \frac{\partial\bar\rho ww}{\partial z} = -\frac{\partial p}{\partial z} + \bar\mu\nabla^2 w + \bar\rho g sin\emptyset + F_{\sigma z} \tag{13}$$