Energy 101 (2016) 229–238

Contents lists available at ScienceDirect

Energy

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

Investigation on the dynamic characteristics of the counter-current flow for liquid desiccant dehumidification

Hao Lu, Lin Lu[*], Yimo Luo, Ronghui Qi

Renewable Energy Research Group (RERG), Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

A R T I C L E I N F O

Article history:
Received 1 July 2015
Received in revised form
16 December 2015
Accepted 5 February 2016
Available online 24 March 2016

Keywords:
Liquid desiccant dehumidification
Counter-current flow
Numerical simulation

A B S T R A C T

The dynamic characteristics of solution film flow with counter-current air flow, especially the two phase contact area, are important for the optimal design and operation of LDD (liquid desiccant dehumidifier). However, the unsteady interfacial information, including the interfacial pressure, velocity and film thickness, is hardly to be examined accurately by theoretical prediction or experimental measurement due to its complexity. In this study, CFD (computational fluid dynamic) models for the counter-current flow of the LDD were established based on the VOF (volume of fraction) and RNG (Renormalization group) k-ε turbulence model. Experimental research had been conducted to validate the liquid film thickness and typical interfacial waves obtained in simulation. With the established model, the dynamic formation process of unsteady counter-current flow is evaluated. Moreover, the effects of various parameters on the liquid film waves, pressure drop, liquid film thickness and interfacial information were obtained and analyzed by the built models. With the increase of liquid flow rate, the roll waves appear and the wave amplitudes are increased significantly, which may obviously enhance the performance in the LDD absorption and regeneration process due to the increased contact area.

1. Introduction

The LDAC (liquid desiccant air-conditioning) system is a promising technology for energy conservation and environmental protection by absorbing the extra air moisture independently. According to the literature [1], LDAC system can save about 10–30% of energy compared with the conventional vapor compression system. If solar energy or waste energy is utilized for desiccant regeneration, the saving can even reach up to 40–50%. The two critical components of LDAC are dehumidifier and regenerator. The falling film flow driven by the gravity force with counter-current air flow is a popular and promising way for the LDD (liquid desiccant dehumidification) because of its low pressure drop and low possibility of droplets carried by air [2,3]. However, the falling film gas–liquid flow is much complicated as it is influenced by many factors such as surface tension, turbulence, shear stress, viscous force, gravity force and the interaction between air flow and desiccant solution. The falling film type can also provide larger contact area for better heat/mass transfer [4]. The characteristics and mechanisms of the counter-current flow are crucial and essential by determining the heat/mass performance of liquid desiccant dehumidifier [5–7]. Therefore, a better understanding on gas–liquid flow in the liquid desiccant dehumidifier is important and meaningful for engineering design and performance improvement.

It has been quite challenging to measure the interfacial information accurately such as pressure, velocity and film thickness by experiments [8–13]. Numerical models, including the finite difference model [14], e-NTU (effectiveness NTU) model [15,16] and some other simplified solutions [17], have also been developed to investigate the gas–liquid flow and the performance of LDAC system. However, many assumptions have been made in these models, such as the falling film flow being steady, fully laminar flow, and the gas–liquid interface being smooth. And the assumptions in the most existing numerical models will result in great deviation with realistic condition. It has been found that the falling film flow pattern is highly associated with Reynolds number of solution [18]. For solution Reynolds number $Re_s < 20$, the falling film flow is laminar and the interface is smooth. For $20 < Re_s < 4000$, the flow become partially turbulent and the interfacial waves appears. Finally, the film flow is fully turbulent and the interface returns to smooth for $Re_s > 4000$. Thus, the falling

---

[*] Corresponding author. Tel.: +852 34003596.
E-mail address: vivien.lu@polyu.edu.hk (L. Lu).

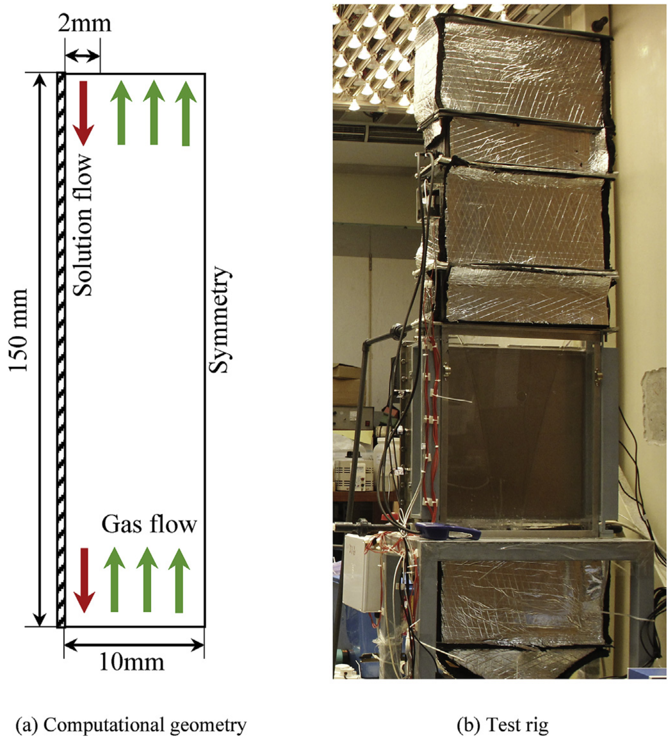
http://dx.doi.org/10.1016/j.energy.2016.02.023

film flow becomes partially turbulent and unsteady even when the solution Reynolds number $Re_s > 20$ [18,19]. Moreover, in practical liquid desiccant systems, the film surface is not smooth and has many small waves due to turbulence and interaction between the liquid film and air flow, which significantly increase the contact area between the air and the solution and further influence the system performance [20]. Therefore, the characteristics and mechanisms of the counter-current flow with interfacial waves and turbulence need to be carefully investigated, yet most of present models did not consider the flow conditions in the dehumidifier.

Recently, the numerical simulation based on the CFD (computational fluid dynamic) models has become an effective way to investigate gas–liquid flow and has successful applications in the evaporators and absorption towers [21,22]. Gu et al. [23] investigated the hydrodynamics of falling film flow on inclined plates by the VOF (volume of fluid) method. They found that the surface tension is crucial on the falling film pattern. Banerjee [24] studied the heat and mass transfer characteristics of the liquid–gas flows in an inclined channel. Ganapathy et al. [25] proposed numerical models for simulating the condensation heat transfer and flow characteristics of two-phase flow in microchannel. Moreover, CFD modeling for falling film over fully wetted horizontal round tube and a finned structure were conducted by Qiu et al. [26] and Mortazavi et al. [27] respectively. Nonetheless, there are limited studies of CFD simulation for LDAC system. In the present study, a numerical model based on CFD is established for simulating the unsteady counter-current flow for falling film liquid desiccant dehumidifier, in which the film waves and turbulent flow are considered. As the counter-current flow is unsteady, partially turbulent and the film thickness is quite thin, it is quite difficult to obtain the whole and accurate interfacial information such as interfacial velocity, pressure and film thickness for experimental study. However, the information is important and can be investigated by suitable CFD model [28–30]. Therefore, in the present study, the numerical model was established to investigate the mechanism of film wave formation as well as the influencing factors on the wave characteristics. The model was verified by the Nusselt empirical formula in terms of film thickness. In addition, the experiments were also conducted to validate the numerical simulation results in the aspect of film thickness and surface structure. Then the model was employed for the analysis of the liquid film waves under different Re_s and different air velocities. The effects of various parameters on the liquid film waves, pressure drop, liquid film thickness and interfacial information were obtained and analyzed by the built models.

2. Numerical model and methodology

In this study, unsteady gas–liquid flows with free interface in a channel between two flat plates are simulated for liquid desiccant dehumidification process. The simplified two-dimensional (2D) geometry model of falling film gas–liquid flow is shown in Fig. 1 (a). The calculation domain is 150 mm high and 10 mm wide, which is in small size for low computation cost. The LiCl solution flows downward along the plate wall from the top inlet with 2 mm wide, while the air flows upward from the bottom inlet, as shown in Fig. 1 (a). The test rig was set up to validate the CFD results, as shown in Fig. 1 (b). The size of the channel is 400 × 100 × 500 mm ($L \times W \times H$). The test surface was a panel made of 316 steel, 400 mm wide (W) and 500 mm long (L). It was equipped with transparent organic glass walls, allowing the viewing of the liquid film. For the experimental test, the JDC-2008 ACCUMEASURE INSTRUMENT developed by Tianjin University was used to measure the film thickness, with the resolution of 0.1 μm and the accuracy of 0.8 μm. A camera was used to record the images of the liquid film (resolution of 3264 × 2448), and some flow patterns were recorded using the video (120 fps). More details for the experiments can be found in the literature [30]. Here, the experimental results are only used to validate the numerical simulation study, as the real interfacial information such as velocity and pressure is quite difficult to be measured in experiments.



(a) Computational geometry (b) Test rig

Fig. 1. Simplified physical schematic of the counter-current flow for LDD.

2.1. Governing equations

There are two key issues in the CFD simulation of air/solution two-phase flow: the solution of gas/liquid turbulence and interface tracking. For the solution of air/liquid turbulent flow, the turbulence model is important for accurate simulation of two-phase flow fields. Mirzabeygi and Zhang [31] compared the performance of different turbulence models (*k-ε model, RNG k-ε model, Realizable k-ε model, k-ω model and k-ω SST model*) on gas–liquid two phase flow simulation with experimental data. The results showed that RNG *k-ε* model and *k-ω SST* model can predict the two phase flow fields more accurately and reasonably. Moreover, Banerjee and Isaac [32] numerically investigated stratified liquid–gas flows by standard *k-ε* model, RNG *k-ε* model and RSM model. The simulation results were compared with experimental data. They found that the RNG *k-ε* model agrees with the experimental results better. Besides, Luo et al. [29,30] successfully simulated the falling film gas–liquid flows with RNG *k-ε* turbulence model. Therefore, the RNG *k-ε* turbulence model is employed in this project for its accurate prediction for complex flow. For interface tracking, the VOF (volume of fluid) model can simulate two immiscible fluids and capture interface well by tracking the volume fraction of each phase.

A series of differential Reynolds averaged Navier–Stokes conservation equations, including continuity, momentum and energy equations, were solved by FVM (finite volume method) with CFD software ANSYS FLUENT. The governing equations for air/solution flow are given as follows [33],