Applied Thermal Engineering 94 (2016) 256–265

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Numerical and experimental investigations of electronic evaporative cooling performance with a coiled channel

Xiang Yin [a], Feng Cao [a,*], Lei Jin [a], Bin Hu [a], Pengcheng Shu [a], Xiaolin Wang [b]

[a] School of Energy and Power Engineering, Xi'an Jiaotong University, 28 Xianning West Road, Xi'an, 710049, China
[b] School of Engineering and ICT, University of Tasmania, Private bag 65, Hobart, TAS 7001, Australia

HIGHLIGHTS

- Two-phase flow cooling performance of a moving electronic unit was studied.
- The distributions of the vapor and the liquid phase in a 3D snake pipe were simulated.
- An experimental setup was established to verify the simulation results.
- The heat concentration in the curve of the channel affects the cooling effect directly.
- An optimized method was developed to improve the overall cooling performance.

ARTICLE INFO

Article history:
Received 12 July 2015
Accepted 22 October 2015
Available online 3 November 2015

Keywords:
Electronic evaporative cooling
Numerical simulation
Coiled channel
VOF
Phase distribution
Cooling performance

ABSTRACT

Numerical and experimental investigations on two-phase evaporative cooling performance for a moving electronic unit with high heat flux have been performed in a coiled channel heat exchanger. Based on the VOF model, the phase distribution of a 3D coiled channel was investigated. It was found that the main vapor streamline flowed through the inner side of one bend channel to the next one, which increased the contact frequency of the vapor and the liquid, improving the cooling performance. Moreover, the results were also verified by the experiment. The cooling effect was directly affected by heat concentration in the bend channel. An optimized method was implemented for this phenomenon. Decreasing the effective flow area and increasing the local velocity could establish a new force balance for the vapor and liquid, eliminating the heat concentration and improving the overall cooling performance.

1. Introduction

With the development of human civilization, electronic equipment sets, such as computers, radios, data servers and electric vehicles, intrude all respects of our daily lives. Some particular electronic equipment sets with smaller size have to deal with a large heat flux. Most of them need to work in relatively stable conditions with a small temperature fluctuation range. When the temperature exceeds certain values, the performance of the components goes down sharply, eventually stopping the device from working. Therefore, the cooling performance of electronic components is always an interesting topic for researchers. However, traditional cooling systems can no longer satisfy the requirements for the cooling performance of electronic components because of their limited cooling capability. Nowadays, two-phase evaporative cooling systems are widely applied in the electronic cooling field, e.g. cooling of radio [1], computers [2,3], or chips [4,5]. To improve its application, the two-phase boiling performance should be further studied via experiments or numerical simulations.

Extensive experimental literature reports on evaporative cooling are available, while only a few deal with evaporative cooling in the macroscopical evaporator by using numerical methods.

In a boiling flow, both the convective boiling and the nucleate boiling regimes affect heat transfer. Different mechanisms constitute the main regime in different flows with different shapes [6]. Aixiang Ma et al. [7] concluded that the inner geometry of evaporators has little influences on the two-phase boiling heat transfer when heat loads are high [8]. Moreover, the Reynolds number has a direct effect on heat transfer coefficient of forced convective boiling [9]. In addition, vapor friction has a little influence on the heat transfer coefficient, while the heat flux and pressure have a stronger influence on the heat transfer [10]. That is to say, the effect of inner geometry on the boiling heat transfer is determined by the application and channel structure. A. Feldman et al. [11] studied the mechanism of boiling heat transfer and figured out that the heat transfer coefficient is independent of mass velocity and quality in the convective regime, but is affected by mass velocity and quality in the nucleate regime. In summary, the above mentioned papers

* Corresponding author. Tel.: +86 29 82663583; fax: +86 29 82663583.
E-mail address: fcao@mail.xjtu.edu.cn (F. Cao).

http://dx.doi.org/10.1016/j.applthermaleng.2015.10.127
1359-4311/

mainly showed that the two-phase cooling system could be better employed in the cooling of electronic components. Nevertheless, much research is still needed in view of better applications, especially in larger electronic units.

Experimental studies on boiling flow are sometimes limited by practical conditions. It is hard to optimize the structure of the evaporator channel, if the boiling performance cannot meet the requirements. Thus, numerical simulations can be an alternative way to solve the problems. Some efforts in the development of two-phase simulations, using the VOF model, have been done to trace the phase interface in a boiling flow. Woorim Lee et al. [12,13] successfully employed the VOF model to analyze the effect of cavity diameter and surface modification on boiling heat transfer. Moreover, numerical simulations have been widely used to investigate the heat transfer performance on the microstructure in a boiling flow [14,15]. The diameter and the number of cavities, as well as the structure of channels, play different roles in different boiling flows [16,17]. The numerical method could be well used to develop the surface configuration for boiling flows. At larger length scales, Christian Kunkelmann and Peter Stephan [18] conducted a numerical investigation on boiling of HFE-7100 according to the experiment of Wanger et al [19]. They found that the departure diameter of a bubble is 2.3 mm, and the results are around 20% above the values obtained experimentally. However, theirs was a 2-dimensional simulation, which could not be directly applied into practice. It is worth pointing out that Y.W. Kuang et al. [20] simulated the flow boiling behavior in 3-dimensional pipes, whose diameters were 32 mm and 65 mm, respectively. The model could well simulate the flow pattern evolution, and capture the hydrodynamic as well as thermal mechanisms. But the geometric model is just a straight pipe. In addition, Z. Yang et al. [21] simulated the boiling flow in a coiled tube, and the results were in good agreement with the phenomenon observed in experiments. It is reasonable to claim that in those works the VOF model could be widely used to analyze the phase distribution. However, they did not analyze the correlation between the heat transfer performance and the phase distribution. As a whole, in the field of boiling flow, a number of investigations have been conducted with the VOF model, including the behavior of the bubble, the evolution of the flow pattern, and the performance of heat transfer. However, most of them are limited to a 2-d scale or to an ideal model. Actually, the influence of the bend on flow pattern or heat transfer of two-phase evaporative cooling has been neglected in the past [22]. The numerical investigation of the bend will be highlighted in this paper.

In the present paper, an attempt was made to investigate the cooling performance difference of a coiled channel and the vapor behavior, which may affect the cooling performance. A numerical simulation, using the VOF model, and the corresponding experiments were conducted to study the evaporative cooling performance for electronic components in a horizontal coiled heat exchanger with a rectangular channel, which could increase the effective heat transfer area. The heat flux of every component ranges from 60 to 120 W/cm^2. The phase distribution was also studied. Numerical analyses of the cooling performance were verified by experimental data directly and indirectly. Moreover, the channel structure was optimized.

2. Simulation method

2.1. Model of simulation

The tracking of phase interface, in the VOF model, was accomplished by solving the continuity equations for the volume fractions of different phases. When the number of the cells is large enough, the errors due to the difference between curves and straight lines could be neglected. The straight line could replace the smooth interface of the two-phase (see Fig. 1). To avoid truncation errors, which would cause a non-conservation of mass velocity between inlet and outlet, the mesh step-size must be selected to match the vapor size. However, in a transient simulation, the time-step is much sensitive to the mesh step-size and must be decreased to confirm stability and convergence, which usually cost much more time. In particular, in 3-dimensional simulations, the simulation period would be too long. Therefore, the mesh step-size should be increased at the expense of simulation accuracy, if the non-conservation is less than 5%. That is to say, some small-sized vapors are neglected.

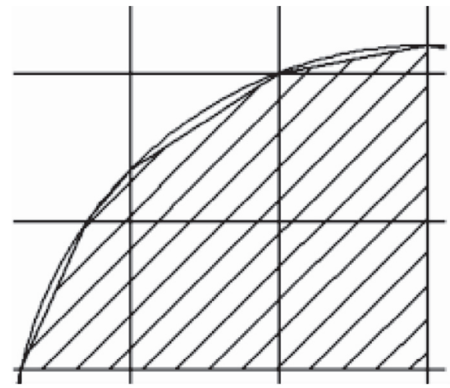

Fig. 1. Interpolation of interface in VOF.

2.2. Governing equations

The fluid in the control volume is governed by three conservation laws: conservation of mass, conservation of momentum and conservation of energy. In the VOF method, the summation of the volume fraction of liquid and vapor equated to 1.

The conservation of mass is:

$$\frac{\partial \alpha_l}{\partial t} + \nabla(\vec{v}_l \cdot \alpha_l) = \frac{S_m}{\rho_l} \quad (1)$$

$$\frac{\partial \alpha_g}{\partial t} + \nabla(\vec{v}_g \cdot \alpha_g) = \frac{S_m}{\rho_g} \quad (2)$$

where S_m is the mass source term that reflects the mass transfer between the two phases in a boiling flow, and l, v represent liquid and vapor, respectively.

The conservation of momentum is:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla\cdot(\rho\vec{v}\vec{v}) = -\nabla \mathrm{p} + \nabla\cdot\left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^T\right)\right] + \rho\vec{g} + \overrightarrow{F_{total}} \quad (3)$$

The force, F_{total}, is the sum of all forces that affect the generation and movement of vapor, including the surface tension, the force between two phases and so on, excluding gravity. Here, the surface tension is highlighted because of phase changes. In a CSF model [23], the surface tension contributes to surface pressure, generating a force that affects the momentum of the fluid. The force follows that

$$F_g = \sigma k \frac{\nabla\rho}{\rho_l - \rho_g} \frac{\rho}{1/2(\rho_l + \rho_g)} \quad (4)$$

where k is the curvature of the interface, formed by the liquid and the vapor. It is defined as

$$k = \frac{\Delta\rho}{|\nabla\rho|} \quad (5)$$