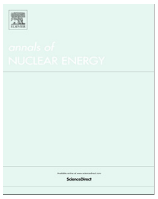
Annals of Nuclear Energy 75 (2015) 158–167

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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



Simulation of boiling flow in evaporator of separate type heat pipe with low heat flux

Y.W. Kuang, Wen Wang *, Rui Zhuan, C.C. Yi

School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 19 February 2014
Received in revised form 2 August 2014
Accepted 5 August 2014
Available online 28 August 2014

Keywords:
Spent fuel pool
Flow boiling
VOF
Numerical simulation
Separate heat pipe

ABSTRACT

The separate type heat pipe heat exchanger is considered to be a potential selection for developing passive cooling spent fuel pool – for the passive pressurized water reactor. This paper simulates the boiling flow behavior in the evaporator of separate type heat pipe, consisting of a bundle of tubes of inner diameter 65 mm. It displays two-phase characteristic in the evaporation section of the heat pipe working in low heat flux. In this study, the two-phase flow model in the evaporation section of the separate type heat pipe is presented. The volume of fluid (VOF) model is used to consider the interaction between the ammonia gas and liquid. The flow patterns and flow behaviors are studied and the agitated bubbly flow, churn bubbly flow are obtained, the slug bubble is likely to break into churn slug or churn froth flow. In addition, study on the heat transfer coefficients indicates that the nucleate boiling is the dominant mechanism in large pipes at low mass and heat flux, with the heat transfer coefficient being less sensitive to the total mass flux.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

Following the alarming catastrophic Fukushima nuclear disaster in 2011, there has been a sudden upsurge of concerns related to the inherent safety of nuclear reactors. On the basis of the conventional pressurized water reactor (PWR), the generation III advanced nuclear reactor AP1000 has dramatically improved its inherent safety features by adopting the concept of passive safety system (Sutharshan et al., 2011). However, the AP1000 does not have passive cooling system for the spent fuel pool (SFP). Thus far, previous studies on SFP have mainly focused on the safety evaluations, with limited attentions on the improvements of effective cooling system of SFP. In the wake of Fukushima accident, there has been increased research interest on the design of passive cooling system for SFP. From the safety perspectives of SFP, it would be ideal to have a separate type heat pipe for cooling. For example, the evaporation section of the heat pipe can be placed around the pool, while the condensation section that is cooled by air under natural convection heat transfer can be installed outside the auxiliary building. The heat source in the pool is the water, which is at a temperature of around 60–90 °C, whereas the heat sink is the environmental air, which is at a temperature of around 10–35 °C (Ye et al., 2013). Obviously, the minimum temperature difference between the pool water and the surrounding air is of the order of 25 °C. Accordingly, the temperature difference between the heat pipe and heat source can be much smaller, making the heat pipe work in low flux.

Generally, a heat pipe consists of an evaporator, an adiabatic section and a condenser. Heat is absorbed by the working fluid in evaporator, which is subsequently transported to condenser, wherein the working fluid condenses and releases the latent heat to the cold media. Following that, the condensate returns to the evaporator through the adiabatic section due to the effect of gravity. In principle, the evaporator has three sections, namely, the liquid phase section, two-phase section and vapor section. Here, the behavior of the fluid in the two-phase section dramatically affects the total heat exchange capacity of the heat pipe. In the case of heat pipes working in high heat flux, the two-phase section can be very long due to the existence of slug and annular flow. Nevertheless, for the heat pipes operating in low heat flux, the slug/annular flow may not exist, whereas the length of liquid section might increase. Therefore, optimization of the design and operation of heat pipes demands deeper insights on the flow behavior and thermal performance of the heat pipe working in low heat flux.

During the last decades, numerous experimental and numerical investigations have been carried out to analyze the thermal characteristics and flow patterns in flow boiling. The typical air–water two-phase flow patterns encountered in conventional vertical tubes are bubbly flow, slug flow, churn flow and annular flow. Most

---

* Corresponding author. Tel.: +86 21 34206096; fax: +86 21 34206814.
E-mail address: wenwang@sjtu.edu.cn (W. Wang).

http://dx.doi.org/10.1016/j.anucene.2014.08.008
0306-4549/© 2014 Elsevier Ltd. All rights reserved.

Nomenclature

A	area [m^2]
E	energy [J/kg]
f_c	condensation coefficient [–]
f_e	evaporation coefficient [–]
F_{vol}	volumetric force [Pa/m]
g	gravity acceleration [m/s^2]
G	mass flux [kg/(m^2 s)]
h	heat transfer coefficient [W/(m^2 K)]
h_{fg}	latent heat [J/kg]
Δh	grid size [m]
k	thermal conductivity [W/(m K)]
M^*	mass flux at interface [kg/(m^2 s)]
m	mass transfer term [kg/(m^3 s)]
M	mass per mole [kg/mol]
Na	number of active nucleation sites per unit area [/m^2]
p	pressure [Pa]
q	heat flux [W/m^2]
R	universal gas constant [J/(mol K)]
S	source term [–]
t	time [s]
T	temperature [K]
u	velocity [m/s]
v	specific volume [m^3/kg]
V	volume [m^3]

Greek symbols

α	volume fraction
ρ	density [kg/m^3]
μ	dynamic viscosity [kg/(m s)]
σ	surface tension coefficient
κ	surface curvature

Subscripts

b	bulk flow
cell	cell
h	heat
inter	interface
l	liquid phase
v	vapor phase
out	out side
sat	saturation
w	wall

of the flow patterns tend to occur in the two-phase flow with evaporation, while the slug flow and annular flow are expected to have higher heat transfer coefficients. In the past decades, flow pattern maps obtained from air–water experiments in vertical tubes have been widely used to describe the flow behavior. However, these flow pattern maps are not suitable for illustrating the flow boiling of refrigerant (Hashizume, 1983). To this end, Zürcher et al. (2002) improved adiabatic flow pattern map on the basis of the data of hydrocarbon refrigerants (HFC-134a and HFC-407C) and ammonia. Nevertheless, due to Taylor instability, conventional slug flow cannot sustain itself in a large vertical tube (Cheng et al., 1998; Kytömaa and Brennen, 1991). Rather, the unstable slug bubble is more likely to breakup resulting in an increase of turbulence and recirculation. In general, large pipes often encounter bubbly flow, churn slug flow and churn froth flow. In other terms, the flow patterns in large pipes are much different from the conventional patterns. Factually, during the evolution of flow patterns in boiling flow, it is highly imperative to include the contributions of mass transfer between phases, in addition to the considerations on Taylor instability. In addition, influence of turbulence also plays a significant role in the development of boiling flow. It is therefore necessary to gain a comprehensive understanding of the flow patterns with heat transfer, if the heat pipe works in low heat flux, as this will affect the prediction of two-phase flow in the separate type heat pipe designs used in SFP cooling system.

Computational fluid dynamics (CFD) modeling with the algorithm of either volume of fluid or level-set is considered to be an effective method for the investigations of two phases flow. For instance, Yang et al. (2008) reported a numerical simulation of flow boiling with R-141B in a horizontal coiled tube. The numerical predictions of flow behavior were found to be in good agreement with the experimental observations. On the other hand, the flow profiles of the simulation reported by Ghorai and Nigam (2006) correspond to the open literature, which uses a RNG k–ε model besides VOF. Meanwhile, Končar et al. (2004) developed a two-fluid model of sub-cooled nucleate boiling flow in a vertical channel. The simulated and experimental results were found to be in good agreement, although flow patterns were not observed. Furthermore, on the basis of the VOF model, Zu et al. (2011) proposed the pseudo nucleate boiling in which the bubbles are generated via injecting vapor through the inlet located on the wall. The 3-D simulation in the rectangular micro channel reproduces the experimental observations of the distorted profile of the bubble and its trajectory. These studies suggest that the numerical simulation has become a powerful method for investigating the two-phase flow. However, the inherent complexity of two-phase flow necessitates further studies on this phenomenon. In principle, the evolution of two-phase flow pattern is determined by various parameters, including viscosity, surface tension, and momentum exchange between the gas and liquid, in addition to gravity. Moreover, under conditions of phase change, the mass and momentum transfer between the two phases cannot be neglected.

The newly proposed SFP cooling system consists of heat pipes of diameter 65 mm, working with ammonia. The surface tension of ammonia is approximately five times lesser than that of water. Consequently, the flow dynamics and thermal performance in the evaporation section of the heat pipe working in low heat flux may be different from conventional pipes. Therefore, experimental data obtained from conventional pipes with slug flow may not be suitable for understanding the flow structure and thermal characteristics. In addition, the visualization experiment for the 65-mm heat pipe seems to be unrealistic and uneconomical. Hence, a CFD modeling including flow boiling and turbulence seems to be a pragmatic solution for predicting the fundamental flow mechanism in the 65-mm heat pipe.

To this end, in the present work, we have made an attempt to develop a flow boiling model, wherein a 3D numerical simulation of ammonia flow boiling in the evaporation section of the heat pipe was conducted to understand the two-phase flow behavior. The flow patterns in the evaporation section were studied. Meanwhile, a comparison of heat transfer coefficients obtained by simulation and empirical correlation has also been presented. The numerical results obtained in this study are expected to be beneficial for improving the design of the heat pipe in passive SFP.

Download English Version:

https://daneshyari.com/en/article/8068955

Download Persian Version:

https://daneshyari.com/article/8068955

Daneshyari.com