Contents lists available at ScienceDirect



International Journal of Heat and Mass Transfer

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

## Numerical investigation on a separated structure shell-and-tube waste heat boiler based on experiment



IEAT and M

### Huaishuang Shao, Qinxin Zhao\*, Zhiyuan Liang, Yungang Wang

Key Laboratory of Thermo-Fluid Science and Engineering, Ministry of Education, Xi'an Jiaotong University, Xi'an 710049, Shaanxi, China

#### ARTICLE INFO

Article history: Received 17 April 2017 Received in revised form 11 September 2017 Accepted 29 September 2017

Keywords: Shell-and-tube waste heat boiler Vapor-liquid two-phase flow Numerical simulation Experimental validation

#### ABSTRACT

In the present paper, a novel separated structure shell-and-tube waste heat boiler is investigated through both experimental and numerical methods. The whole numerical model based on a scaled experimental facility is developed to analyze the shell-side thermal-hydraulic performances. Instead of using isothermal boundary condition on the tube wall, a linear heat flux is alternative according to the temperature difference measured in the experiment. The Drift-Flux model is used to solve the vapor-liquid twophase flow across tube bundle. The commercial software ANYSY Fluent is adopted to conduct numerical computation, and a considerably good accuracy is obtained by comparing with experiment. The pressure drop, void fraction, flow field and heat transfer characteristic of two-phase mixture flow boiling across tube bundle are presented and analyzed in detail, respectively. The results show that the maximum pressure loss occurs at the inlet of risers, which can severely weaken the overall natural circulation. The HTC of heating tubes on the shell-side is mainly determined by void fraction rather than velocity inside the tube bundle, as the latter changes little along flow direction. This work can be as a reference for designers and operators.

© 2017 Elsevier Ltd. All rights reserved.

#### 1. Introduction

Shell-and-tube heat exchanger (STHX) has been widely used in the industrial fields. One of the most common designs is the shelland-tube waste heat boiler, which is used to recover waste heat energy from industry exhaust. Two flow domains are divided by a horizontal tube bundle placed in a shell, which the heating flue gas flows inside the tubes while the heated water boils on the outside surface of the tubes. In order to improve the ability and efficiency of the boiler, the separated structure shell-and-tube waste heat boiler (SS-STWHB) has been developed for decades. A steam dome with several risers and downcomers is configured above the shell to separate vapor from saturated water, as showed in Fig. 1(a). More tubes can be placed in the shell for the purpose of increasing flue gas flow and power. Meanwhile, the diameter of the shell can be made smaller to ensure reliable operation under the same heat load. The difference in density between the twophase mixture flowing within the tube bundle and the liquid water flowing in the downcomer makes natural circulation occur. However, the two-phase mixture flow boiling in the shell is a complex 3D problem. Fluid moves vertically from the outlet of the

\* Corresponding author. *E-mail address: zhaoqx@xjtu.edu.cn* (Q. Zhao). downcomers to the inlet of the risers, as well as axially and horizontally in cross sections perpendicular to the shell axis, as showed in Fig. 1(b). The flow instability and local overheating often occurs in the actual operation of the process. Analysis and optimization of two-phase mixture flow boiling in the shell are important for the design and safe operation of the SS-STWHBs.

Experiment study is highly reliable and precise to provide the measurement of thermal-hydraulic performances which include pressure drop and heat transfer characteristic. However, experiment is resource-wasting and time-consuming. The flow characteristics of two-phase flow field inside a tube bundle are also hard to obtain intuitively through the experiment. Nowadays, a lot of research works about the STHXs have been conducted with the help of computational fluid dynamics (CFD). Many researchers prefer to optimize and design the STHXs using CFD because of the obvious benefit on economic cost, flow path observation and time-consuming [1]. A brief review on the numerical simulation study is as follows.

At first, the flow distribution inside a heat exchanger has been obtained through the solution of the continuum Navier-Stokes equations. In 1974, Patankar and Spalding [2] first proposed the idea of using distribution resistance to simulate the presence of heat-transfer tubes and baffle plates on the shell side of a heat exchanger. The porous media model was improved by AbuRomia

Nomenclature			
C d f <sub>drag</sub> g h k L m	coefficient diameter drag function gravity constant enthalpy turbulent kinetic energy latent heat mass flow rate	$ \Gamma $ $\varepsilon $ $\lambda $ $\mu $ $\rho $ $\tau $ $\Phi_l^2 $ $Xtt$	evaporation/condensation rate turbulent dissipation effective conductivity coefficient viscosity density relaxing time multiplier parameter
n P Pr R, V Re S t U V X Y, G	constant pressure Prandtl number parameter Reynolds number source term time error velocity vapor mass fraction generation of turbulence kinetic energy	Subscri c dr e eff g l m p t	ipts condensation drift evaporation effective gas liquid mixture particle thermal
Greek symbols α void fraction			



Heating tube

(a) Overall appearance diagram

(b) Two-phase flow on the shell-side

Fig. 1. Separated structure shell-and-tube waste heat boiler description.

et al. [3] and Sha et al. [4], and the idea of accounting for the anisotropy using surface permeabilities was introduced. Prithiviraj and Andrews [5] developed a 3D control volume using the concept of distributed resistance to simulate the flow and heat transfer in the STHXs. Compared with experimental data, good simulation results of overall pressure drop and temperature distribution were obtained. He et al. [6] analyzed the fluid flow and heat transfer characteristics of shell-side fluid in the STHXs using the porous model. A modified k- $\varepsilon$  model coupled with the distributed resistance method was used to account for the effects of tubes on turbulence generation and dissipation. Shi et al. [7] carried out a semi-porous media approach to simulate airflow through largescale sparse tubular heat exchangers. The results showed that this numerical approach could reduce computational cost significantly and offer a reasonable agreement with experiment data. In addition to the porous model, the unit model and the periodic model are also used by many researchers in the numerical simulations of the STHXs.

For the unit model, the flow zone enclosed by tubes is regarded as a unit flow channel. The impacts of inlet, outlet and shell wall on the fluid flow characteristics are ignored. Dong et al. [8] performed a numerical investigation on the rod baffle heat exchanger using the unit model. The results were compared with correlations as well experiments. A good agreement was obtained, even though the inlet and outlet nozzles effects were not taken into account. Sheikholeslami [9-14] studied the nanofluid forced convective heat transfer using unit model to obtain the results. Many parameters, such as nanofluid volume fraction and Darcy, have been discussed through the numerical method. You et al. [15] applied the unit model to investigate the shell-side hydraulics of the STHXs with trefoil-hole baffles. The relative deviations of both convective heat transfer coefficient and pressure drop between numerical results and their experiments were within 5%. For the periodic model, the inlet and outlet are set as periodic boundary conditions. The effects of inlet and outlet on the shell-side are also ignored. Zhang et al. [16] conducted the simulation for a heat exchanger

Download English Version:

# https://daneshyari.com/en/article/7054789

Download Persian Version:

https://daneshyari.com/article/7054789

Daneshyari.com