Cryogenics 80 (2016) 115–126

Contents lists available at ScienceDirect

Cryogenics

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

Research paper

Numerical simulation of coupled fluid flow and heat transfer characteristics in a submerged combustion vaporizer

Chang-Liang Han, Jing-Jie Ren, Yan-Qing Wang, Wen-Ping Dong, Ming-Shu Bi *

School of Chemical Machinery, Dalian University of Technology, No.2 Linggong Road, Ganjingzi District, Dalian, Liaoning 116024, PR China

ARTICLE INFO

Article history:
Received 10 June 2016
Received in revised form 2 October 2016
Accepted 5 October 2016
Available online 5 October 2016

Keywords:
Submerged combustion vaporizer
Liquefied natural gas
Coupled heat transfer
Numerical simulation

ABSTRACT

Submerged combustion vaporizer (SCV) has been widely employed in Liquefied Natural Gas (LNG) receiving terminals as the main peaking-shaving facility. In the current work, numerical simulation was carried out to study the intrinsic fluid flow and heat transfer characteristics inside the SCV. After the verification of the numerical model and method with the experimental data, detailed results about flow field and temperature field were presented to have an understanding of the principle of shell-side heat transfer enhancement. The distributions of local LNG temperature, wall temperature, water bath temperature and heat transfer coefficient along the tube length were also revealed. Moreover, the influences of main operating parameters such as flue gas temperature, inlet LNG velocity, inlet LNG pressure, static water height and flue gas flux on the system performance were systematically investigated. Finally, based on current simulation results, two empirical correlations were proposed to predict the coupled heat transfer performance of SCV. The simulated results could provide some insight into the design and optimization of SCV.

1. Introduction

Submerged combustion vaporizer (SCV) is a classic multiphase reactor, which has achieved the success in the vaporization of Liquefied Natural Gas (LNG) because of its unique advantages in compact structure, quick response to changing process parameters and high thermal efficiency above 99% [1]. Typical SCV system composes of combustion air blower, fuel tank, submerged burner, water tank, weir, flue gas distributor and heat exchange tube bundle, etc., as shown in Fig. 1. In the SCV, the water bath firstly absorbs quantity of heat from flue gas generated by the submerged burner through a great deal of bubbles, and then it transfers heat to the cryogenic LNG inside the tube bundle completely confined by the specific weir. Compared to other vaporizers, SCV involves more complex dynamic behavior, such as two-phase mixture sweeping the stagger tube bundle and trans-critical LNG flowing inside the serpentine tube coils simultaneously. As a consequence, a better knowledge about the coupled fluid flow and heat transfer process inside and outside of tube bundle is necessary to design and improve the overall thermal performance of SCV.

In addition to the SCV, the commonly used vaporizers in the LNG station also cover the ambient air vaporizer (AAV), open rack vaporizer (ORV), superORV and intermediate fluid vaporizer (IFV). The thermodynamics analysis of them had been performed by previous researchers. Han et al. [2] presented a method based on CFD simulation to describe the heat transfer process in a practical SCV with the wall temperature gradient assumption boundary conditions. The influences of initial water height and flue gas flux on the heat transfer behavior outside the tube bundle were investigated. Simulation results illustrated that the Zhukauskas equation could be applied for the prediction of shell-side heat transfer coefficient. Qi et al. [3] established a distributed parameter model concerning a SCV heat transfer tube. However, they simplified the structure of tube bundle as a single serpentine tube. The effects of LNG mass flow rate, inlet LNG temperature and inlet LNG pressure on the water bath temperature were discussed. The results showed that the water bath temperature decreased with increasing inlet LNG temperature, and increased with the increase of LNG mass flow rate, in particularly, it was insensitive to the inlet LNG pressure. Park and Kim [4] carried out the optimization study of SCV using Entropy Minimization Method (EMM) for two representative boundary conditions: constant wall temperature and constant heat flux. The results demonstrated that highly efficient heat transfer surface unit cell for optimizing SCVs could be designed using the EMM analysis. Gavelli [5] developed a CFD-based modeling method to quantitatively predict the formation, dispersion and dissipation of the AAV fog. The results showed an increase in the elevation of discharge would not only lead to the faster mixing the effluent with the air but also shorter fog

* Corresponding author.
E-mail address: bimsh@dlut.edu.cn (M.-S. Bi).

http://dx.doi.org/10.1016/j.cryogenics.2016.10.001
0011-2275/

Nomenclature

H	static water height
K	heat transfer coefficient
A	tube bundle area
Q	heat exchange rate
h	enthalpy
l	tube length
N	number of tube
T	temperature
q	heat flux
m	mass flow rate
D	outer tube diameter
Re	Reynolds number
Pr	Prandtl number
Cp	specific heat
v	velocity
V	volume flow rate
R	diameter of bend tube
P	pressure

Greek symbols

ε_g	gas holdup
ρ	density
λ	thermal conductivity
μ	dynamic viscosity

Subscripts

fg	flue gas
b	water bath
t	tube-side
s	shell-side
w	tube wall
in	inlet
out	outlet
we	weir
pc	pseudo-critical value
cr	critical value
pre	predicted value
cal	calculated value

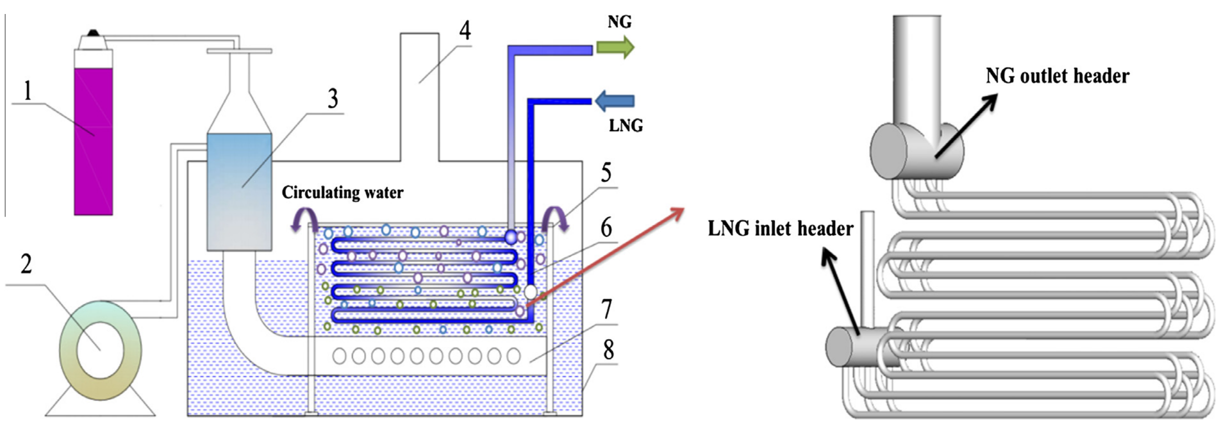

Fig. 1. The typical schematic of a submerged combustion vaporizer and the geometry of heat exchange tube bundle: 1. Fuel tank, 2. Combustion air blower, 3. Submerged burner, 4. Stack, 5. Weir, 6. Heat exchange tube bundle, 7. Flue gas distributor, 8. Water tank.

dispersion distances. Kuang et al. [6] presented a numerical model to investigate the heat transfer performance of AAV. The simulated results suggested the AAV should work for the long time operation under the condition of low inlet mass flow rate to avoid the formation of frost on the vaporizer fin. Jin et al. [7] established a thermal model to simulate the LNG evaporating process inside the tube of superORV. They concluded that the required tube lengths for this type of vaporizer were grown with the increase of ice thickness. An undated model focusing on superORV heat transfer tube was proposed by Pan et al. [8]. The effects of main operating parameters on the heat transfer performance were discussed. The results showed that the technology of heat transfer enhancement could shorten the required minimum tube length by 60%. Xu et al. [9] carried out the numerical investigation to compare the basis heat transfer areas of the IFV using different candidate refrigerants. The results demonstrated that propylene and dimethylether were prospective medium for the IFV. Pu et al. [10] built the thermal model for an IFV, the effects of inlet operating parameters on the IFV performance were systematically studied. Their major findings were the saturation temperature of propane increased with the increase of temperature and mass flow rate of inlet seawater, while it decreased with increased of inlet LNG mass flow rate.

Upon the foregoing review, although numerous investigations have focused on the thermal analysis of LNG vaporizer, few studies have reported the coupled fluid flow and heat transfer characteristics of SCV. In this paper, we attempt to propose the numerical approach to describe the heat transfer process between the two-phase mixture and trans-critical LNG outside and inside of tube bundle. The model and method are firstly verified with our experimental data. Based on the proposed model, the coupled heat transfer characteristic inside the SCV is investigated. And the effects of main operating parameters on system performance are also discussed. Moreover, two Nusselt number correlations are used for the prediction of thermal performance of SCV. The paper is of great importance for the design of efficient SCV.

2. Numerical simulations

2.1. Geometrical model

The computational physical model for the SCV was shown in Fig. 2, which mainly consisted of the water tank, two horizontal serpentine tubes, flue gas distributor and weir. Table 1 listed the