International Journal of Heat and Mass Transfer 68 (2014) 110–118

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

A numerical investigation on LNG flow and heat transfer characteristic in heat exchanger

Handry Afrianto [a], Md. Riyad Tanshen [a], B. Munkhbayar [a,b], U. Tony Suryo [c], Hanshik Chung [a], Hyomin Jeong [a,*]

[a] Department of Energy and Mechanical Engineering, Gyeongsang National University, 445 Inpyeongdong, Tongyeong 650-160, Gyeongnam, Republic of Korea
[b] School of Chemical Engineering, The University of Adelaide, SA 5005, Australia
[c] Department of Mechanical Engineering, Diponegoro University, Semarang, Indonesia

ARTICLE INFO

Article history:
Received 20 December 2012
Received in revised form 15 September 2013
Accepted 16 September 2013
Available online 5 October 2013

Keywords:
CFD
LNG heat transfer
1–2 Shell and tube exchanger
Buoyancy effect

ABSTRACT

The current article reports a numerical study on liquid natural gas (LNG) flow and heat transfer characteristic in heat exchanger at 0.6 MPa pressure condition. The computational fluid dynamics (CFD) code FLUENT was used to simulate the fluids flow and heat transfer characteristic in heat exchanger. The three-dimensional model of 1–2 heat exchangers was used to perform the numerical simulation. The numerical simulation of the current study was validated and compared with a reference data, indicating effectiveness-NTU and LMTD method. Therefore, the properties of LNG were calculated from NIST standard database 4 (SUPERTRAPP) versions 3.2. The variation effects of mass flow rate and heat transfer characteristic of the fluids were investigated. The numerical results show that the heat transfer coefficient increases with increasing the mass flow rate. Furthermore, the optimization of mass flow rate of fluids for vaporization process was reported in this study.

1. Introduction

The liquid natural gas (LNG) is one of the clean energy resources that useful in industry fields, such as power production, district heating; and pipeline gas for households, automotive and maritime industries. Natural gas is converted to LNG by cooling into 111 K at atmospheric pressure. Therefore, the volume reduces until 600 times, which allows keeping natural gas in storage tank. Recently, the new technology has been developing to use LNG as a maritime fuel to diminish emission effect, followed by global warming issue because of the LNG has a low emission when used as ship fuel. The LNG has been saved in fuel storage tank in liquid condition. Therefore, LNG converts to natural gas before its use as ship engine fuel using vaporization devices (heat exchangers). In the heat exchanger, the LNG adsorbs of heat and it changes to gas due to passing through the pipe, as shown in Fig. 1. Pipe is heated by hot water from the heat transport between the electric heaters in reserve tank.

However, LNG reduces the heat transfer efficiency of the system due to cryogenic condition of LNG, vaporization occurs in frost formation. The other problems are non-uniformity thermal properties, buoyancy force resulting from non-uniform density distribution over the cross section of the channel, and acceleration or deceleration of the flow. These are due to expansion or contraction of the fluid as a result of significant axial variations of bulk temperature under heating or cooling, as reported in He et al. [1]. Therefore, in respect of effective design of vaporization device, flow and heat transfer characteristic are very important to investigate. To design the heat exchanger device, there are many parameters need to be considered according to internal structure and mechanism of heat transfer complicatedly related to design parameters, as reported by Kara and Guaras et al. [2]. Noie et al. [3] was employed the experimental and theoretical solution to investigate the thermal performance of air-to-air thermo-syphon heat exchanger. A computer simulation program is based on the effectiveness-NTU method, which was developed to estimate the outlet temperature and the experimental result was close to those obtained from computer simulation and became better as the velocity increases.

Thantrung et al. [4] was compared numerical simulation and experimental tests were to study the fluid flow and heat transfer characteristics for a rectangular shaped heat exchanger using ε-NTU method. The result obtained from numerical analyses was a good agreement with that obtained from experimental data of the heat transfer coefficient estimated to be less than 9%. Hasan et al. [5] evaluated the effect of channel geometry on heat transfer and fluid flow behaviors of counter-flow heat exchanger using numerical simulation. The effectiveness and performance indexes were expressed in terms of relative size of channel; Reynolds number and thermal conductivity ratio of the solid wall to that of the

* Corresponding author. Tel.: +82 55 640 3184.
E-mail address: hmjeong@gnu.ac.kr (H. Jeong).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.09.036

Nomenclature

A	area, m^2
Bo	buoyancy force, kg/m^2
Cp	heat specific, J/(kg K)
d	tube diameter, m
f	fouling coefficient
g	gravitational acceleration, m/s^2
h	heat transfer coefficient, $W/(m^2 K)$
k	thermal conductivity, W/(m K)
l	tube length, m
N	number of tube
NTU	number of transfer unit
Pr	Prandtl number
Q	heat exchange rate, W
Re	Reynolds number
T	temperature, K
U	overall heat transfer coefficient, $W/(m^2 K)$
ΔT	temperature different, K

Greek letters

β	thermal expansion coefficient
ε	effectiveness
$\dot{m}$	mass flow rate, kg/s
μ	dynamic viscosity, kg/(m s)
ρ	density, kg/m^3
υ	kinematic viscosity, m^2/s

Subscripts

Av	average value
b	bulk value
max	maximum
min	minimum
i	inside
o	outside
s	shell side
t	tube side

fluid. Mathew et al. [6] studied theoretical approach for application of effectiveness-NTU relationship to parallel flow heat exchangers with non-dimensional parameters such as temperature, heat transfer rate and axial distance.

In the present study, the LNG flow and heat transfer characteristic in heat exchanger were numerically investigated. The main aim of the present study was to investigate the flow and heat transfer characteristic of LNG in heat exchanger using numerical simulation. In this study, we applied the LNG and water with initial temperature values was 110 and 323 K, respectively. The CFD commercial code of Fluent was employed to solve the problem by means of the finite volume method. Therefore, the numerical simulation has been validated and compared with the reference data from experimental study, indicating the effectiveness-NTU and LMTD method. The LNG material property was assumed to calculate from NIST standard database 4 (SUPERTRAPP) versions 3.2. The numerical results show the heat exchanger model was achieved in temperature distribution, optimization of mass flow rate of fluids for vaporization, heat transfer coefficient and buoyancy effects.

2. Descriptions and numerical procedure

The computational model of shell and tube heat exchanger is shown in Fig. 2, and the geometry parameters are listed in Table 1. Model was consisting of steady flows and heat transfer of fluids inside shell and tube heat exchanger. At the inlet condition, a stable temperature and variation of mass flow rate prevail. The working fluids in the tube and shell side were liquid natural gas (LNG) and water (H_2O), respectively.

2.1. Governing equations

A numerical approach was used to predict of LNG flow and heat transfer characteristic in heat exchanger, the general governing equations are as following:

Conservation of mass:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

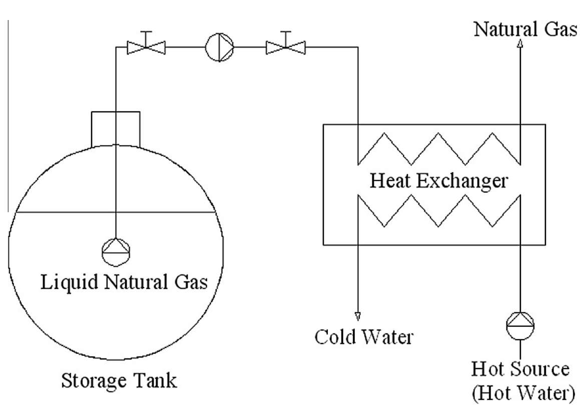

Fig. 1. Schematic of LNG vaporization system.

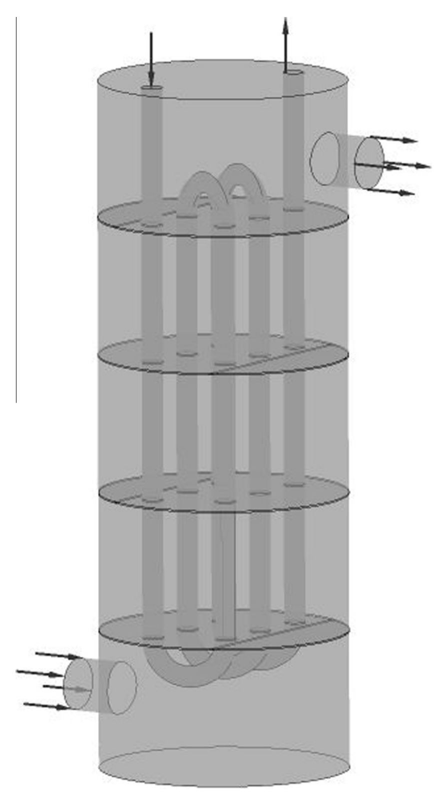

Fig. 2. Geometrical configuration under study.