Desalination 413 (2017) 109–118

Contents lists available at ScienceDirect

Desalination

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

A numerical model and experimental verification for analysing a new vacuum spray flash desalinator utilising low grade energy

A. Hosseini Araghi [a,*], M. Khiadani [a], M.H. Sadafi [b], K. Hooman [b]

[a] School of Engineering, Edith Cowan University, 270 Joondalup Drive, Joondalup WA 6027, Australia
[b] School of Mechanical and Mining Engineering, University of Queensland, Brisbane, St Lucia QLD 4072, Australia

HIGHLIGHTS

- Vacuum spray flash desalinator is a core component of the new DTECD technology.
- This research provides thermodynamic and CFD models to simulate the vacuum desalinator.
- Deviation of vapor-liquid equilibrium model is 5% for estimating the evaporation rate.
- Deviations of DPM to predict the temperature field and evaporation are 9% and 17%, respectively.
- A DPM approach is developed to predict the thermo-fluid behaviour of the vacuum spray desalinator.

ARTICLE INFO

Article history:
Received 3 August 2016
Received in revised form 2 March 2017
Accepted 8 March 2017
Available online xxxx

Keywords:
Desalination
DTECD
Vacuum spray flash evaporation (VSFE)
CFD simulation
DPM model

ABSTRACT

This study investigates the performance of a new vacuum spray flash desalinator, a core component of the open water cycle in a discharge thermal energy combined desalination (DTECD) technology using theoretical and experimental techniques. The feedwater contains 3.5wt% of NaCl while the inlet temperature range can vary over a range of 55 °C to 75 °C based on the low temperature utilised in the DTECD system. In order to design an efficient desalinator, physical aspects of the proposed vacuum spray flash evaporation (VSFE) should be studied. Thus, an experimental study was undertaken to verify the theoretical evaporation rate and centreline temperature data. The proposed desalinator was modelled using a CFD model implemented in the available package ANSYS FLUENT 16.2 and some results are compared with a thermodynamic model embedded in ASPEN/HYSY 8.0. It was observed that the defined thermodynamic models based on vapor-liquid equilibrium in the Aspen and Fluent can predict the evaporation rate with the average errors of 5% and 17%, respectively. Moreover, discrete phase model (DPM) approach can analyse the thermo-fluid field in the desalinator with acceptable accuracy about 9%. Droplets size, velocity, temperature and concentration profiles are predicted and the underlying physics are discussed regarding the VSFE geometry.

1. Introduction

Desalination is an important technology to provide potable water for many areas around the world. Due to the high amount of energy consumption required for water treatment, development of the new desalination technologies depend on alternative energy sources such as renewable and waste energies [1]. The new design of some desalination technologies are relied on the using of low grade energy for evaporation of saline water such as ocean thermal energy desalination (OTED) or other low temperature thermal desalination (LTTD) plants [2–5]. These new desalinators are not comparable with mega capacity conventional desalination plants regarding the production rate, but they are very economical to use as a heat recovery co-generator in combination with energy intensive industries.

Discharge thermal energy combined desalination (DTECD) is a new combined desalinator, which recovers waste heat from an energy intensive industry and cogenerates freshwater and power [6,7]. DTECD technology has two subsystems: closed power cycle and open water cycle as shown in Fig. 1. In addition, the core component of the open water cycle is a single-stage vacuum spray flash evaporation (VSFE) drum. Furthermore, the proposed system introduces a new configuration of a spray desalinator and utilises a seawater ejector for vacuum generation, which leads to a more energy-efficient design compared to similar technologies [6,7]. The overall efficiency of the proposed cogeneration-heat recovery system is about 50% and its operation does not depend on fossil fuels [8,9].

* Corresponding author.
E-mail address: a.hosseiniaraghi@our.ecu.edu.au (A. Hosseini Araghi).

http://dx.doi.org/10.1016/j.desal.2017.03.014
0011-9164/

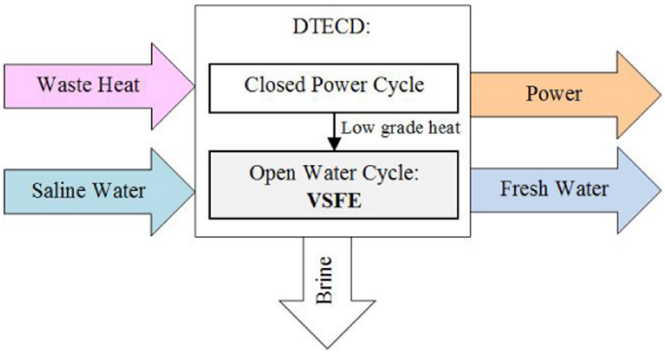

Fig. 1. Conceptual block diagram of DTECD core technology: VSFE desalination system.

The key component in the new LTTD process is a single-stage VSFE drum and it has already reported that how its performance affects significantly the overall performance of the DTECD system [8]. The main reason to use spray flash evaporation phenomena in a desalinator is based on the Myateke et al. [10] report, which revealed that spray flash evaporation demonstrates higher performance compared to a pool flash evaporation. Previous experimental studies provide data relevant to the single-stage flash desalination system and analysed evaporation rate [11], heat and mass transfer phenomena [12,13], thermodynamic approach and thermal performance [8,14–16]. Also, Ikegami et al. reported that an upward spray shows higher heat transfer performance compared with a downward spray [17]. However, these limited experiments are not sufficient to describe the related numerical approach and design specifications of a proposed VSFE system.

On the other hand, there are some articles about optimizing the spray flashing processes using a CFD approach: spray dryer with a downward flow [17–19], spray dry cooling tower at an atmospheric condition [20], pressurized spray brine evaporator [21], spray distillation column [22], and spray fuel combustion chamber [23–25]. While these studies have proven useful, they do not provide comprehensive evaluation of thermo-fluid behaviours that would be needed to investigate this new vacuum desalinator with different configuration, operating and evaporation conditions. In addition, to the author's knowledge, CFD analysis of the proposed vacuum spray flash desalinator has not previously been performed. Hence, this research presents two simulation approaches to determine the impacts of key parameters: a CFD model based on an Euler-Lagrange discrete phase model is used to investigate droplet transport phenomena and a thermodynamic model based on the electrolyte-non-random-two-liquid (ENRTL) equation is used to analyse the thermal equilibrium of the system. The numerical simulations, using for evaporation rate, droplet size reduction, and centreline temperatures are validated against the experimental data. The experiment is performed as three different tests based on DTECD temperature range, which is between 55 °C and 75 °C. Finally, some thermo-fluid behaviours of the system are predicted and discussed. The base case for the last part is at 65 °C of injected feed to drum under 6 kPa vacuum pressure.

2. Models and equations

2.1. Conservation laws and droplet equations

The equations describing the conservation of mass, energy and momentum to predict the flow field and can be solved numerically. Behaviour of droplets as a dispersed phase in a continuous phase is simulated by a discrete phase model. Firstly, the compatibility of the standard Navier-Stokes equations in the FLUENT under vacuum conditions should be checked by calculating the Kundson number.

The Kundson number can be used to determine the flow regime in a specific vacuum device by defining the degree of gas rarefaction [26]. It is specified as the ratio of the mean free path to a local spatial scale given by Eq. (1).

$$\mathrm{Kn} = \frac{\lambda}{\mathrm{L}} \tag{1}$$

For Kn ≤0.1, continuum model of the Navier-Stokes equations implemented in a CFD model describes precisely the gas flow behaviour [27]. In contrast, the Navier-Stokes equations cannot describe the rarefied gas flow for Kn >0.1 [28].

One of the issues with spray flash evaporation simulation is the coupling of conservation equations between the droplets and the continuous phase. Fig. 2 shows how the mass conservation, momentum exchange and energy conservation, from droplet to the continuous phase, are respectively coupled with evaporation, drag and heat transfer equations [29]. The related conservation equations are defined as below [30,31] using the Eulerian-Lagrangian approach.

The mass conservation equation for the continuous phase is:

$$\frac{\partial(\rho_C)}{\partial_t} + \nabla\cdot(\rho_C U_C) = \Gamma_{m,CD} \tag{2}$$

The conservation equation of component "i" is obtained by:

$$\frac{\partial(\rho_C y_{iC})}{\partial t} + \nabla\cdot(U_C y_{iC}) - \nabla\cdot(\rho_C D_{iC}(\nabla y_{iC})) = \Gamma_{m,iC} \tag{3}$$

Source term in the mass conservation equation is defined using the droplet evaporation rate as [32]:

$$\Gamma_{m,iC} = \frac{dm_D}{dt} = -\pi\phi_D\rho_C D_{iC} Sh(x_{iD}K_e - y_{iC}) \tag{4}$$

The momentum equation is represented by:

$$\frac{\partial(\rho_C U_C)}{\partial_t} + \nabla\cdot(\rho_C U_C U_C) = -\nabla p + \nabla\cdot\tau + \rho_C g + \Gamma_{mo,CD} \tag{5}$$

where, τ is the viscous stress tensor which takes the following form for a Newton fluid:

$$\tau = \mu_C\left[\nabla U_C + (U_C)^{\tau}\right] \tag{6}$$

Besides, the interphase momentum transfer is based on interfacial forces exerted on the two phases and presented as:

$$\Gamma_{mo} = F_{CD}^{drag} + F_{CD}^{lift} + F_{CD}^{wall} + F_{CD}^{virtual\ mass} + F_{CD}^{turbulent\ dispersion} \tag{7}$$

where, the subscripts of C and D are continues and discrete phases, respectively.

In this research only the interfacial drag force is considered according to Ropelato et al. assumption for modelling a distillation column [22]. The drag force used for dispersed phase flowing in a continuous phase [32], is given by:

$$F_{CD}^{drag} = \sum \frac{3C_{drag}\rho_C}{4\rho_D\phi_D}(U_C - U_D)^2\dot{m}_D\Delta t \tag{8}$$

where, C_{drag} is described as:

$$C_{drag} = \frac{24}{Re}\left(1 + 0.15\,Re^{0.687}\right) \tag{9}$$

The energy conservation equation is represented by:

$$\frac{\partial(\rho_C h_C)}{\partial t} + \nabla\cdot(\rho_C U_C h_C) - \nabla\cdot(k_t\nabla T_C) = \sum_{i=1}^{Ncomp}\Gamma_{i,mC}h_C + Q_C + \Gamma_{e,CD} \tag{10}$$