Desalination 339 (2014) 40–53

Contents lists available at ScienceDirect

Desalination

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

Numerical analysis of curved vane demisters in estimating water droplet separation efficiency

G. Venkatesan [a,*], N. Kulasekharan [b], S. Iniyan [c]

[a] NIOT, Chennai 600 100, Tamil Nadu, India
[b] Department of Mechanical Engineering, Saveetha Engineering College, Chennai 602 105, Tamil Nadu, India
[c] Institute for Energy Studies, Anna University, Chennai 600 025, Tamil Nadu, India

HIGHLIGHTS

- 2D RANS simulations conducted for predicting demister vane separation efficiency
- DPM approach with Rosin–Rammler distribution is used for water droplet injection.
- Wave model is suitable among droplet breakup models available in the solver.
- Inclusion of Saffman lift force does not influenced the results much.
- SA model or Realizable k–ε model can be considered for turbulence closure.

ARTICLE INFO

Article history:
Received 15 December 2013
Received in revised form 9 February 2014
Accepted 12 February 2014
Available online 5 March 2014

Keywords:
2D RANS
Two-phase flows
Water droplets
Curved vane demister
CFD

ABSTRACT

The present work deals with the numerical investigations of vaned demisters using a computational fluid dynamics tool to validate the presently adopted computational methodology. A vane demister profile identified from literature, on which experimental data is available, is chosen and water particles at the rated flow rate and distribution are injected. Discrete phase modeling approach with Rosin–Rammler size distribution is chosen for specifying the water droplet injection with a combination of break-up and/or coalescence option, along with few turbulence model recommended in the literature are tested for suitability. The walls of the demister are set to capture the water droplets impinging on them and will be removed from the numerical calculations. Various options within the solver are numerically tested to recommend an appropriate combination of solver setting. The results indicate that the Wave model is the best option among several other droplet breakup models available in the solver. It was observed that inclusion of Saffman lift force does not influence the results much. Spalart–Almaras model or Realizable k–ε model can be considered as a suitable turbulence closure for numerical prediction of vane separation efficiency in curved vane demisters.

1. Introduction

Desalination is one of the prime sources for fresh water generation from seawater. Both quality and quantity of fresh water generated per day depend on the technology being employed and the usage like drinking, agricultural or industrial purposes. Difficulty was faced in maintaining the salinity levels in the fresh water generated. Hence the need arises to remove the entrained water droplets by using components like demisters. These include packing, filters, settling tanks, electrostatic precipitators, cyclones, baffles, vanes and wire mesh demisters. Choice of selection from these devices depends on the composition of the mist or droplet, size, flow rate, temperature, and other operating conditions.

Vane mist eliminators or demisters find wide applications since they do not clog easily and offer lesser resistance to flow than other types. As vanes are less susceptible to re-entrainment and flooding, these are more effective at higher velocities and handle higher droplet sizes while the wire mesh demisters are more suitable for removing smaller particles at lower velocities. Vane type demisters have high vapor handling capacity than wire mesh types which can be utilized in chambers operating at wide range of pressure and temperature. Hence the size of the flashing equipment can be reduced considerably.

Wave-plate mist eliminators generally consist of parallel blades or vanes spaced to provide passage for vapor flow and profiled with angles to provide sufficient change of direction for liquid droplets to impact, coalesce and drain from the surfaces of the plates. The wave or vane mist eliminators are typically designed for the gas to enter a narrow path in between the appropriately shaped plates and the flow has to follow the plate geometry. Normally, when the vapor and entrained

* Corresponding author. Tel.: +91 99624 26689 (cell).
E-mail addresses: gvniot@gmail.com (G. Venkatesan), cfdsekar@gmail.com (N. Kulasekharan), iniyan777@hotmail.com (S. Iniyan).

http://dx.doi.org/10.1016/j.desal.2014.02.013

liquid droplets pass through a demister, the vapor moves freely through the demister pad but the liquid droplets, due to their greater inertia, cannot make the necessary sharp turns. As a result the droplets are thrown into contact with the plate surfaces and briefly held there by impinging on the plates. As more droplets enter the pad and collect on the surfaces, they grow in size, run down to the bottom surface of the plate separator and drain and fall from the unit. Thus the entrained liquid is removed from the gas flow effectively, usually by inertial impingement. The airflow carries the droplet but the effect of droplets on the airflow is negligible. The liquid droplet will be separated when the gravity force is larger than the drag force of the up flowing gas stream. Drainage slots on vane surface also help to allow liquid to disengage from gas stream.

Few of these drops when re-entrained into the stream at higher vapor velocities, lead to reduced droplet collection efficiency. To overcome this problem, hooks are introduced at locations where the deposited liquid may accumulate and droplet re-entrainment is reduced or prevented. The hooks help to capture the droplets and air velocity is higher at the cross section where hook is affixed, the separator shows high efficiency even though the inlet velocity is low. The presence of the hook promotes the formation of eddies which separates the fine droplets, but also result in greater pressure drop due to their strong turbulence dissipation. Water droplet separation efficiency and pressure drop across the demister are the primary parameters of interest in a desalination process, to assess the performance of the demisters. Measuring droplet distribution and number of droplets before and after the demister vanes to assess its droplet separation efficiency using experiments is costlier and also needs more skill and expertise. In order to achieve a tradeoff between the separation efficiency and pressure drop for different vane demister geometries, and to choose an optimal combination of parameters, numerical simulation studies need to be carried out. Several computational studies have been carried out by several researchers in the past, to find the influence of various geometrical parameters and numerical choices on the performance prediction of flow through straight vane demisters.

Wang et al. [1] conducted Computational Fluid Dynamics (CFD) studies using a commercial CFD code to carry out a comprehensive numerical investigation on droplet removal efficiency and pressure drop of straight vane demister by varying the parameters like inlet gas velocity, bend angle and rear pockets. Standard k–ε turbulence model was used to simulate the gaseous phase and turbulent dispersion effects on droplet trajectories were not taken into account. However, no comparison with experimental data was reported. Galletti et al. [2] performed a numerical simulation for gas flow and droplet motion in a straight vane demister with drainage channels using Shear Stress Transport (SST) turbulence model and compared the result of their simulation with the experimental results of Ghetti [3]. The results indicated that the SST turbulence model gives better results than standard k–ε model.

James et al. [4] conducted numerical simulations of the gas flow and liquid droplet dispersion, coupled with mathematical models of the film deposition and separation. Commercial CFD software CFX with standard k–ε model was used to predict the primary turbulent flow field and a separate eddy-interaction model was then used to predict the liquid droplet dispersion and deposition for a variety of droplet size distributions at inlet to the eliminator. They attempted to provide a method of determining whether, under a given liquid loading, re-entrainment takes place.

Jia et al. [5] studied the droplet behavior in the wave-type flow channels, to understand the breakup of droplets by impingement on liquid film to form secondary droplets. Realizable k–ε model was applied with standard wall function for wall node treatment, when the hydraulic Reynolds number was above 4000. The suggested numerical method showed the pressure drop and separation efficiency in good agreement with the experimental data. But secondary droplet generation from droplet breakup was not discussed in detail.

Rafee et al. [6] studied the droplet transport and deposition in the turbulent airflow inside a wave-plate mist eliminator, using an Eulerian–Lagrangian computational method. The Reynolds Stress Transport Model (RSTM) with standard wall functions and with enhanced wall treatment was used for simulating the airflow field using a code developed in-house. Their results showed that the enhanced wall treatment improved the predictions of the droplet removal efficiency especially for small droplets. On the other hand, the RSTM with standard wall functions cannot predict the removal efficiency correctly, especially for low gas velocities.

Josang et al. [7] made an experimental investigation on a curved vane demister to measure the air velocities in strategic locations. Rafee et al. [8] carried out numerical simulations based on Eulerian–Lagrangian method and compared their results with the experimental data of Josang et al. [7], where the turbulent droplet laden air flow inside a single passage of a curved vane demister is reported. Their results show that by including the wall reflection terms in transport equations of the Reynolds stresses, better predictions can be achieved than those obtained by RSTM without wall reflection terms. The distribution of water droplets, its generation and deposition mechanisms using the Phase Doppler Anemometry (PDA) technique is discussed in detail by Josang [9]. A brief summary of droplet generation mechanisms including droplet–droplet interaction, droplet break up, splashing of impinging droplet and re-entrainment from liquid film is reported. Numerical simulation of air flow carrying water droplets through the curved demisters is also carried out and the results are compared with the experimental data.

A brief survey of literature indicated that the experimental and numerical studies with droplet transport and deposition reported are mostly on wave type or straight vane type demisters and the research work on curved vane demisters is found to be very little. The numerical studies reported so far did not give a clear recommendation of a particular turbulence model and a choice of droplet breakup models for the prediction of flow through curved vane demister. Hence a numerical study is conducted to estimate the performance of a curved vane demister, using chosen turbulence models identified from literature and from the previous work by the present authors [10] and droplet breakup models available within the commercial CFD code Ansys Fluent 14.0.

Computations are performed on the domain used by Josang [9] on which the experimental measurements were available. The results from the present computations were compared with the experimental data. The primary objective of the present work is to identify suitable options in turbulence closure; discrete phase modeling approach, droplet breakup model and inclusion of other options like Saffman's lift force and Stochastic collision along with the default droplet breakup option. Hence the boundary conditions are chosen from the literature values and the computational results are validated with the experimental results. The present study is focused on the multi-phase flow of air carrying water droplets to study the droplet entrainment, carryover and flooding in the curved vane demister and a future work is planned to include droplet evaporation and condensation.

2. Physical and computational model

2.1. Physical geometry of the demister

To study the influence of chosen turbulence models and the droplet break-up mechanisms available in the CFD solver and to validate the present computational methodology, the experimental dataset of Josang [9] on curved vane demister geometry is chosen. The demister geometry on which the experiments were conducted is shown in Fig. 1 whose geometry profile details are available. The vane used in the experiments has an axial length of 171 mm in the x-direction, a transverse pitch of 25 mm in the y-direction and a depth of 108 mm in the z-direction. The ratio of vane depth in the z-direction to the