International Journal of Heat and Mass Transfer 81 (2015) 618–637

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Three-dimensional numerical model for the two-phase flow and heat transfer in condensers

Pooya Mirzabeygi, Chao Zhang [*]

Department of Mechanical and Materials Engineering, Western University, London, Ontario N6A 5B9, Canada

A R T I C L E I N F O

Article history:
Received 25 July 2014
Received in revised form 8 October 2014
Accepted 9 October 2014
Available online 18 November 2014

Keywords:
Heat transfer
Computational Fluid Dynamics (CFD)
Condenser
Porous medium
Gas–liquid two-phase flow
k–ε turbulence model
Closure correlations
Inundation

A B S T R A C T

In this study, a three-dimensional numerical model has been developed to simulate the fluid flow and heat transfer in industrial steam surface condensers with complex irregular shapes. The numerical method is based on the Eulerian–Eulerian two-phase model by solving the conservation equations of mass and momentum for both gas-phase and liquid-phase, and the mass fraction conservation equation for the non-condensable gases. A distributed resistance formulation (porous media) approach is used to account for the effect due to the presence of tubes on the shell-side flow in the condenser. Also, the convective heat transfer correlations are used to model the heat and mass transfer between the shell-side fluid and the tube-side fluid. The effect of the turbulence and non-condensable gases on the primary phase flow is also included in the numerical model. The numerical results are compared with the experimental data for a small experimental condenser and a full-size industrial condenser, and the proposed numerical model is proved to be relatively accurate in simulating the turbulent two-phase flows in condensers. Finally, the comparison is made between the current three-dimensional model and the quasi-three-dimensional model proposed in the previous work.

1. Introduction

Condensers are commonly used in industrial applications. Most of them contain banks of thin-walled tubes and the steam condenses over or within these tubes through which the heat transfer occurs with the coolant. The condensate liquid formed during this process then falls under gravity into the condenser storage well [1].

In this study, the focus is on steam surface shell-and-tube condensers which are commonly used in power plant industry to condense the exhaust steam from the turbine to liquid water. The mixture of the steam and non-condensable gases (mainly air) flows in the shell-side and the condensation occurs outside of horizontal tubes through which the cooling water flows. There might be one or more partition plates used in the condenser to support the weight of the tube bank depending on the condenser and tubes design characteristics. These tube supports provide an air-tight fit, so that the shell-side fluid cannot pass through them.

The design of condensers has traditionally been conducted by experienced designers through trial and error approach based on previous designs, or tests [2–5]. However, a detailed knowledge of the flow field and heat transfer in condensers is required in order to design a reliable and efficient unit, and the traditional methods do not provide this level of details. One way to overcome this issue is to perform experimental tests; however, this type of experiments are usually expensive and time consuming [6]. Also detailed turbulence measurements and flow visualizations are difficult to perform if not impossible. Therefore, a reliable, robust numerical model can be a useful tool to provide further insight to the complex flow and heat transfer happening in an industrial condenser.

Several numerical models have been proposed to simulate flow and heat transfer in condensers. Patankar and Spalding [7] were pioneers in this field as they introduced the porous media concept to account for the drag force due to tube bundles, and consequently, the computational time can be significantly reduced as the numerical procedure does not require the detailed analysis of flow and temperature fields around each tube. Many numerical models have been developed based on the porous media concept with a reasonable accuracy [8–28]. These models can be classified into two main categories: single-phase and two-phase models.

In single-phase models, the effect of the second phase is neglected and it is assumed that the condensate disappears from the domain as it is formed. Davidson and Rowe [8] solved the single-phase equations and used Berman and Fuks correlation to account for the effect of non-condensable gases. Al-Sanea et al.

---

[*] Corresponding author. Tel.: +1 519 661 2111.
E-mail addresses: pmirzabe@uwo.ca (P. Mirzabeygi), czhang@eng.uwo.ca (C. Zhang).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.10.015
0017-9310/

Nomenclature

a, b, $c2$	constants
A	heat transfer area (m^2)
$C_{1\varepsilon}$, $C_{2\varepsilon}$	constants in turbulence transport equations
$C_{\varepsilon b}$	coefficient
C_{fx}, C_{fy}	interphase friction coefficient
C_{phase}	interphase exchange coefficient
C_P	specific heat (J/K)
D	diffusivity of air in steam (m^2/s)
D	diameter (m)
f	friction factor due to tube bundle
f_d	friction factor due to interphase friction
f_R	Darcy friction factor
G_k	generation of turbulent kinetic energy (kg/m s^3)
g	gravitational acceleration (m/s^2)
h	heat transfer coefficient W/(m^2 K)
J	diffusion flux of the air in the vapor (kg/m^2 s)
k	turbulent kinetic energy (J/kg)
L	latent heat of condensation (J/kg)
$\dot{m}$	condensation rate (kg/s)
n	inundation index
Nu	Nusselt number
p	Pressure (Pa)
P_t	tube pitch (m)
Pr	Prandtl number
R	thermal resistance (m^2 K)/W
Rb	source term due to tube bundle
RG	source term for RNG k–ε (kg/m s^3)
Re	Reynolds number
S	general source term
S_{mass}	continuity source term (kg/m^3 s)
S_{mom}	momentum source term (N/m^3)
S_{diff}	species transport source term (kg/m^3 s)
T	temperature (K)
U	velocity magnitude (m/s)
u	x-velocity (m/s)
V	velocity vector (m/s)
v	y-velocity (m/s)
V_L	volume of the computational cell (m^3)
Wb	source term due to interphase friction (N/m^3)

Greek symbols

α	porosity
β	volume fraction
Γ	effective diffusivity (Pa s)
γ_c	condensation rate in the control volume (kg/s)
γ_{tot}	condensate leaving the control volume (kg/s)
ε	turbulent kinetic energy dissipation rate (m^2/s^3)
ϕ	air mass fraction
λ	thermal conductivity (W/m K)
μ	laminar viscosity (Pa s)
ξ	pressure loss coefficient
ρ	density (kg/m^3)
σ	turbulent Prandtl number
τ	time scale
$\bar{\bar{\tau}}$	stress tensor (Pa)
ω	specific dissipation rate (1/s)

Subscripts

a	non-condensable-gas (air)
c	liquid condensate
ci	condensate film interphase
cw	coolant water
d	droplet
eff	effective
g	gas-mixture
id	inside diameter
k	parameter for turbulent kinetic energy
l	liquid
od	outside diameter
ow	tube outside wall
p	particle
rel	relative
s	steam
tw	tube wall
ε	parameter for turbulent kinetic energy dissipation rate

[9], Malin [14], Roy et al. [15], Prieto et al. [16], Ormiston et al. [17,18], and Zhang et al. [26] were among the first researchers who performed the simulations of condensers using single-phase models, in which the effects of the inundation and interphase interaction on the fluid flow were neglected. Zhang and Zhang [28] studied the sensitivity of the condensation heat transfer coefficient using a single-phase model. They studied the effects on the heat transfer rate due to the inundation, inlet air mass fractions, and cooling water flow rate using a single-phase model. He et al. [2] performed a three-dimensional single-phase analysis using a modified k–ε model for shell-and-tube heat exchangers. Nedelkovski et al. [29] developed a finite element method to solve three-dimensional single-phase equations for the condenser where the effect of non-condensable gases was considered. Later, Rusowicz [30] performed the simulation using a single-phase two-dimensional steady state model on a 50 MW plant condenser and reported satisfactory results. Zeng et al. [4] solved the single-phase mixture equation for the mixture of steam and non-condensable gases in addition to the RNG k–ε model to study the effect of different tube arrangements for a 300 MW power plant condenser.

Two-phase models regard both gas and liquid as continuous and interpenetrating fluids, and the interaction between the phases are modeled. The first model was developed by Al-Sanea et al. [10], in which they included the interphase effect in their two-phase model. Rabas and Kassem [11] included the condensate inundation effect and neglected the vapor shear effect on the convective heat transfer in condensers. McNaught and Cotchin [12] implemented a new correlation to account for the effect of inundation. Bush et al. [13] predicted the flow and heat transfer in an experimental condenser using a two-phase model. Moreover, Zhang and Bokil [24] included the interphase effect and used a two-phase model to simulate a condenser. Hu and Zhang [19] developed a modified k–ε turbulence model for two-phase flows in condensers, and later assessed the effects of different closure correlation on the numerical simulations [20] and proposed a new correlation to account for the effect of the inundation on the convective heat transfer in condensers [21].

In this study, an Eulerian–Eulerian model for the two-phase flow and heat transfer in condensers will be developed alongside a robust turbulence model for the primary phase accounting for the extra turbulence kinetic energy generation and dissipation rate due to the presence of the tube bank and the condensate droplets. The mass and momentum conservation equations for both gas and liquid phases, and the species transport equation for the non-condensable gas are solved. The relevant correlations are selected based on previous studies to model the effect of the condensate inundation and non-condensable gases on the heat and mass transfer in condensers.