International Journal of Heat and Mass Transfer 89 (2015) 229–241

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Turbulence modeling for the two-phase flow and heat transfer in condensers

Pooya Mirzabeygi, Chao Zhang *

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

ARTICLE INFO

Article history:
Received 28 October 2014
Received in revised form 9 May 2015
Accepted 10 May 2015

Keywords:
Heat transfer
Computational fluid dynamics
Condenser
Porous medium
Gas–liquid two-phase flow
Turbulence model

ABSTRACT

In this paper, various turbulence models are compared to determine which models are most accurate in simulating two-phase fluid flow and heat transfer in steam surface condensers. The numerical method is based on the conservation equations of mass and momentum for both gas-phase and liquid-phase, and mass fraction conservation equation for non-condensable gases. Modified standard k–ε and RNG k–ε models are proposed to account for the effects of tube bundles and two-phase interactions on the gas-phase turbulence. A quasi-three-dimensional approach is used to account for the effect of the temperature difference in the coolant flowing in the tubes. Finally, the numerical results are compared with the experimental data to assess the performance of the proposed turbulence models.

1. Introduction

Heat transfer between two fluids at different temperatures is vital for most industrial processes, and heat exchangers are the devices that specifically designed for this purpose. Heat exchangers are widely used in a variety of applications, such as power production, petroleum refineries, chemical plants, food industries, air conditioning, refrigeration, sewage treatment etc. [1]. Among different types of heat exchangers, shell and tube heat exchangers are the most widely used (37% of total heat exchanger market) because of their simple manufacturing and adaptability to different ranges of operating conditions, pressures, and temperatures [2]. A shell and tube heat exchanger consists of a tube bundle mounted inside the shell. One fluid flows over the tubes (shell side) and the other fluid flows through the tubes, and heat transfer occurs between these two streams of fluids. The heat exchanger can operate in a single phase mode to heat up or cool down fluids or it can operate in a two-phase mode as an evaporator or condenser.

The focus of this study is on steam surface condenser which is a type of shell and tube heat exchanger commonly used in power plants to condense the exhaust steam from the turbine to liquid water in order to complete the thermal cycle. In this type of condensers, coolant water flows through the tube bundles to extract heat from the oncoming steam in the shell side. In the shell-side, the mixture of the steam and non-condensable gases flows around the tubes and the liquid condensate occurs. The liquid then falls down due to the gravity and exits the condenser at the bottom.

Many academic and industrial researches have devoted to improve the design of shell and tube heat exchangers in order to increase the thermal efficiency and reduce the energy consumptions. However, it is impossible to develop a reliable and efficient design without a detailed knowledge of the flow field and heat transfer in the heat exchangers. This knowledge can be obtained through numerical or experimental analysis. However, experimental analysis is usually time-consuming and expensive; besides, it is difficult, if not impossible, to perform full scale measurements in an industrial condenser [3]. Moreover, with the introduction of new generation of computational resources that reduces the cost and time of the analysis, numerical models are getting more and more attention from researchers to provide accurate information on the flow and heat transfer in industrial applications. Therefore, a reliable, robust numerical model can be used as an alternative tool to provide further understanding into what is happening in a full scale industrial condenser.

Industrial condensers usually contain large number of tubes, and it is therefore practically impossible to solve the detailed fluid flow and heat transfer around each and every tube in the condenser. To solve this problem Patankar and Spalding [4] proposed a method to consider the pressure drop due to presence of tube bundles using porous media analogy. In this method, each computational cell may contain several tubes, and the effect of these

* Corresponding author. Tel.: 1 519 661 2111x88345.
E-mail addresses: pmirzabe@uwo.ca (P. Mirzabeygi), czhang@eng.uwo.ca (C. Zhang).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.05.044
0017-9310/

Nomenclature

a, b, c_2	constants
A	heat transfer area (m^2)
$C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ	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/kg K)
D	diameter (m)
D	diffusivity of air in steam (m^2/s)
D_e	effective diffusivity of air in steam (m^2/s)
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)
x	coordinate (m)
y	coordinate (m)

Greek symbols

α	local porosity
α_t	porosity in the tubular region
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)
θ	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
m	phase indicator
od	outside diameter
ow	tube outside wall
p	particle
rel	relative
s	steam
tw	tube wall
x	component in x-direction
y	component in y-direction
ε	parameter for turbulent kinetic energy dissipation rate

tubes is accounted for by using a distributed resistance against the gross motion of the fluid. Many researchers adopted this approach due to its simplicity, easy implementation and robustness in solving large-scale industrial processes [3,5–9].

Moreover, accounting for the presence of the tube bundle as a distributed resistance inhibiting the motion of shell side flow is not the only challenge in the road to develop a comprehensive numerical model that can accurately solve the flow field and heat transfer process in an industrial condenser. Other challenges are mostly due to the presence of turbulence in both vapor and condensate phases, large number of tubes, presence of non-condensable gases, three dimensionality effects due to the presence of coolant flow in the tubes, condensate inundation etc. Resolving all these complexities to the smallest details is currently not feasible due to limited computational resources. Among these difficulties, turbulence modeling is known to be the most challenging task mainly because there is not a universal turbulence model for multiphase flows that can accurately include the effects of gas–liquid interaction and tube bundles.

Numerical studies of condensers have been conducted by several researchers and different techniques have been proposed to deal with turbulence flows in condensers [10–15]. In most of the earlier researches, the effect of turbulence was neglected [16–21]. Later, a constant viscosity ratio or a simple algebraic equation was used to account for the turbulence [22–24]. Gomez et al. [25] used a length scale (proportional to the clearance between the tubes) and a velocity fluctuation to calculate the effective viscosity. A more recent approach is to solve the turbulence equations for the shell-side flow using different turbulence models to model the turbulent viscosity. Sha et al. [26] proposed a one-equation turbulence model based on transport equation for turbulent kinetic energy to account for tube-bundle resistance. Later, several researchers [3,27–29] solved the k–ε equations to model the turbulent viscosity in the vapor–air mixture, and finally, Zeng et al. [30] implemented the RNG k–ε turbulence model in their numerical solution without considering the effects of tube bundles and two-phase interactions on the turbulence.

However, there has never been a research on comparative analysis of different turbulence models to study the effectiveness of the choice of turbulence models in the simulation of the fluid flow and heat transfer in condensers. Variations of k–ε models, Realizable k–ε [31] and RNG k–ε [32], and also other two equation models such