Applied Thermal Engineering 50 (2013) 445–454

Contents lists available at SciVerse ScienceDirect

Applied Thermal Engineering

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

Three-dimensional performance analysis of plain fin tube heat exchangers in transitional regime

Arafat A. Bhuiyan [a], M. Ruhul Amin [b,*], A. K. M. Sadrul Islam [a]

[a] Department of Mechanical and Chemical Engineering, Islamic University of Technology, Board Bazar, 1704, Bangladesh
[b] Department of Mechanical and Industrial Engineering, Montana State University, 220 Roberts Hall, Bozeman, MT 59717, USA

HIGHLIGHTS

► 3D CFD simulations for plain-fin-and-tube heat exchanger.
► Validated with experimental data.
► Parametric study for the effects of fluid flow and heat transfer.

ARTICLE INFO

Article history:
Received 17 December 2011
Accepted 28 July 2012
Available online 14 August 2012

Keywords:
Heat transfer
Fluid flow
Heat exchanger
Numerical method

ABSTRACT

Three-dimensional CFD simulations are carried out to investigate heat transfer and fluid flow characteristics of a four-row plain fin-and-tube heat exchanger using the Commercial Computational Fluid Dynamics Code ANSYS CFX 12.0. Heat transfer and pressure drop characteristics of the heat exchanger are investigated for Reynolds numbers ranging from 400 to 2000. Fluid flow and heat transfer are simulated and results compared using both laminar and turbulent flow models (k-ω) with steady and incompressible fluid flow. Model validation is carried out by comparing the simulated case friction factor (f) and Colburn factor (j) with the experimental data of Wang et al. [1]. Reasonable agreement is found between the simulations and experimental data. In this study the effect of geometrical parameters such as fin pitch, longitudinal pitch and transverse pitch of tube spacing are studied. Results are presented in the form of friction factor (f) and Colburn factor (j). For both laminar and transitional flow conditions heat transfer and friction factor decrease with the increase of longitudinal and transverse pitches of tube spacing whereas they increase with fin pitches for both in-line and staggered configurations. Efficiency index increases with the increase of longitudinal and transverse pitches of tube spacing but decreases with increase of fin pitches. For a particular Reynolds number, the efficiency index is higher in in-line arrangement than the staggered case.

1. Introduction

Plate fin-and-tube heat exchangers of plain fin pattern are commonly used in the process and HVAC&R (heating, ventilating, air conditioning, and refrigeration) industries. They are compact and light weight. Plain fins are used on the outer surface of the round tubes of staggered or in-lined arrangement to improve the heat transfer coefficient on the gas side in the heat exchangers. The heat transfer between the gas, fins and the tube surfaces is determined by the flow structure. The governing thermal resistance for an air-cooled heat exchanger is usually on the air side which may account for 85% or more of the total resistance [1]. As a result, to effectively improve the thermal performance and to significantly reduce the size and weight of air cooled heat exchangers, the use of enhanced surfaces is very popular in air cooled heat exchangers.

The plain plate fin configuration is the most popular fin pattern, due to its simplicity, durability and versatility in application. During the past few decades many efforts have been devoted to heat transfer and friction characteristics of plain fin-and tube heat exchangers [2–4]. Also a number of correlations are developed by the researchers [5–7]. There are also a number of numerical studies for plain fin-and-tube heat exchangers. Most of the earlier researchers used 2-D and laminar flow conditions in their numerical calculations [8]. These authors noted that 2-D flow field studies cannot sufficiently predict heat transfer between the fluid and the fin; hence their simulations have limited application. Few researchers have reported 3-D modelling for plain-fin configuration

* Corresponding author. Tel.: +1 406 994 6295.
E-mail addresses: arafat@iut-dhaka.edu (A.A. Bhuiyan), ramin@me.montana.edu (M.R. Amin), sadrul@iut-dhaka.edu (A.K.M.S. Islam).

1359-4311/$ – see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.applthermaleng.2012.07.034

Nomenclature

C_P	specific heat at constant pressure, J/kg K
D	tube diameter, m
Fp	fin pitch, m
Ft	fin thickness, m
f	friction factor
H	fin spacing, m
h	average heat transfer coefficient, W/m^2 K
j	Colburn factor
k	turbulence kinetic energy, m^2/s^2
L	flow length, m
Ll	longitudinal tube pitch, m
Lt	transverse tube pitch, m
m	mass flow rate, kg/s
Nu	average Nusselt number
P	local pressure, Pa
P_{in}	inlet pressure, Pa
P_k	share production of turbulence, kg/m s^3
Pr	Prandtl number
Re_H	Reynolds number based on fin spacing, ρuH/μ
T	temperature, °C
T_{in}	inlet temperature, °C
T_{Wall}	wall temperature, °C
u	velocity, m/s
u_{in}	inlet(frontal) velocity, m/s
ε	turbulence dissipation rate, m^2/s^3
λ	thermal conductivity, W/m K
μ	dynamic viscosity, Ns/m^2
μ_T	turbulent viscosity, Ns/m^2
ρ	fluid density, kg/m^3
ω	turbulent frequency, s^{-1}

in their numerical studies [9–12]. The experimental studies have shown that the flow range for the plain-fin configurations extends from laminar to the transitional range [1]. So it is necessary to determine the flow structure for the plain fin configuration for both laminar and transitional flow regions numerically. Panse [13] has done a comparative numerical analysis of the three turbulence models namely κ-ε model, RNG κ-ε model and the k-ω model.

In the present study, 3D CFD simulations have been carried out to investigate the performance for the plain fin configuration for both laminar and transitional flow regions because, most of the early researchers have considered 2D modelling in their numerical analysis. A researcher reported 2-D numerical results along with experimental data for the influence of fin spacing on the heat transfer and pressure drop, Kundu et al. [14]. These researchers were limited, because they were trying for solutions for a three dimensional problem with two dimensional approach. The authors noted in their study that the two dimensional field studies cannot sufficiently predict heat transfer between the fluid and the fin. The approximations used by the authors were to avoid the side wall effects. Zdravistch et al. [15] used different boundary conditions for his study in the two dimensional flow field. The author specifically mentioned the importance of three dimensional simulations in case of side wall effects, which is exactly the case with heat exchangers which use fin and tube banks. It is understandable from the literature that for the flow channel consists of a complicated structure, numerical studies with two-dimensional approach is not sufficient. So for the present study of similar type of geometry and boundary condition, 2D model will not be the appropriate as side wall has significant effect on the performance of heat exchanger performance.

2. Governing equations

The present study was performed considering thermal transport with convective heat transfer. Air is used as working fluid assuming constant properties ($k = 0.0261$ W/m K, $\mu = 1.831 \times 10^{-05}$ Ns/m^2, $Pr = 0.736$, $\rho = 1.185$ kgm^{-3}). Assuming a steady three dimensional incompressible flow with no viscous dissipation and viscous work, effects of body force and buoyancy are neglected; both laminar and transitional flow conditions are considered. For transitional flow solutions three turbulent models are used namely standard k-ε, k-ω and RNG k-ε model. The flow is described by the conservation laws for mass (continuity), momentum (Navier–Stokes) and by the energy equations are as follows:

$$\left[\frac{\partial u_i}{\partial x_i} = 0\right] \tag{1}$$

$$\rho\left(u_j\frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu+\mu_T)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

$$\rho C_P\left(u_j\frac{\partial T}{\partial x_j}\right) = \frac{\partial}{\partial x_j}\left[\left(\lambda+\frac{\mu_T C_P}{Pr_T}\right)\frac{\partial T}{\partial x_j}\right] \tag{3}$$

In Equations (2) and (3) μ_T and Pr_T are turbulent viscosity and turbulent Prandtl number respectively. As suggested by Yuan [16], $Pr_T = 0.9$ was used in the current study. The value of μ_T is determined based on the specific turbulence model that is being used. In k-ω turbulent model the μ_T is linked to the turbulence kinetic energy (k) and turbulence frequency (ω) via the following relation:

$$\mu_T = \rho\frac{k}{\omega} \tag{4}$$

The transport equations for k and ω were first developed by Wilcox [17] and later it was modified by Menter [18], can be expressed as:

$$\rho\left(u_j\frac{\partial k}{\partial x_j}\right) = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_{k3}}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \beta^{'}\rho k\omega \tag{5}$$

$$\rho\left(u_j\frac{\partial \omega}{\partial x_j}\right) = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_{\omega 3}}\right)\frac{\partial \omega}{\partial x_j}\right] + 2(1-F_1)\rho\frac{1}{\sigma_{\omega 2}\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} + \alpha_3\frac{\omega}{k}P_k - \beta_3\rho\omega^2 \tag{6}$$

In Equation (6), F_1 is a blending function and its value is a function of the wall distance. $F_1 = 1$ and 0 near the surface and inside the boundary layer respectively. The constants of this model (Φ_3) are calculated from the constants Φ_1 and Φ_2 based on the following general equation.

$$\Phi_3 = F_1\Phi_1 + (1-F_1)\Phi_2 \tag{7}$$

The model constants are given as $\alpha_1 = 5/9$, $\beta' = 0.09$, $\beta_1 = 0.075$, $\sigma_{\kappa 1} = 2$, $\sigma_{\omega 1} = 2$, $\alpha_2 = 0.44$, $\beta_2 = 0.0828$, $\sigma_{\kappa 2} = 1$, $\sigma_{\omega 2} = 1/0.856$. Details of these different turbulent models are documented in