Applied Thermal Engineering 30 (2010) 2011–2018

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

A first-principles simulation model for the thermo-hydraulic performance of fan supplied tube-fin heat exchangers

Maicon Waltrich [a], Christian J.L. Hermes [b,*], Joaquim M. Goncalves [c], Cláudio Melo [a]

[a] POLO Research Laboratories for Emerging Technologies in Cooling and Thermophysics, Federal University of Santa Catarina 88040-970, Florianópolis-SC, Brazil
[b] Department of Mechanical Engineering, Federal University of Paraná P.O. Box 19011, 81531-990, Curitiba-PR, Brazil
[c] AC & R Center, Federal Institute of Technological Education of Santa Catarina Rua José Lino Kretzer 608, 88103-310, São José-SC, Brazil

A R T I C L E I N F O

Article history:
Received 29 January 2010
Accepted 4 May 2010
Available online 19 June 2010

Keywords:
Tube-fin heat exchanger
Condenser
Gas cooler

A B S T R A C T

This paper outlines a novel first-principles mathematical model to simulate the thermo-hydraulic behavior of compact fan-supplied tube-fin heat exchangers for light commercial refrigeration applications, i.e., with heat duties ranging from 0.5 to 2.0 kW. The model is based on the mass, momentum and energy conservation equations applied to both the refrigerant and air streams. The model predictions were compared with experimental data taken at several operating and geometric conditions. It was found that the model predictions for the air-side heat transfer and pressure drop were very close to the experimental data with maximum deviations of ±10% and ±15%, respectively. The model was employed to assess the thermal-hydraulic performance of a gas cooler running with supercritical CO_2 as working fluid.

1. Introduction

The annual Brazilian energy consumption due to air conditioning and refrigeration equipment is approximately 50,000 GWh, which corresponds to almost half of the Itaipu hydropower plant capacity [1]. Such a figure not only highlights the needs for better energy utilization practices, but also points out the urgency for more efficient refrigeration systems. It is well-known that the energy consumption of refrigeration systems is mostly due to the irreversible thermodynamic losses that take place in each of the system components, among which the heat exchangers present the higher performance/cost ratio.

Most of the heat exchangers used in small-size refrigeration and air conditioning systems are of the tube-fin type, where the air flows externally over extended finned surfaces whereas the refrigerant flows internally along the tubes. Performance analysis of this type of heat exchanger is usually assessed experimentally, thus requiring specific test rigs, prototype samples and trained technical personnel. A faster and less costly alternative is to employ mathematical models to simulate the thermo-hydraulic behavior of the heat exchanger coils. The simulation not only rationalizes the number of prototypes and experiments needed, but also permits the heat exchanger optimization based on both component-level (e.g., j and f) and system-level (e.g., COP) performance indicators.

However, most of the heat exchanger simulation tools available in the literature [2–10] does not take into account the fan-coil hydrodynamic interaction, although recent studies [11–13] have indicated that the fan-coil interaction plays an important role on either performance or cost-driven design processes.

In this context, a mathematical model to simulate the thermo-hydraulic performance of tube-fin heat exchangers considering the fan-coil hydrodynamic interaction is proposed herein. Experiments were also carried out not only to provide the necessary data for the model validation exercise but also to identify the most appropriate heat transfer and pressure drop empirical correlations required by the model.

2. Mathematical formulation

The performance characteristics of tube-fin heat exchangers depend on the air, refrigerant and heat flows over the fins and tubes. These phenomena are governed by the mass, momentum and energy conservation equations, whose solution is complex and computer demanding. In order to balance model accuracy and mathematical complexity, the proposed heat exchanger model was divided into two sub-models namely, thermal and hydraulic. The former provides the heat transfer rate and the thermodynamic states

* Corresponding author. Tel.: +55 41 3361 3239.
E-mail address: chermes@ufpr.br (C.J.L. Hermes).

1359-4311/$ – see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2010.05.006

Nomenclature

Roman

A	heat transfer surface, m^2
A_{face}	heat exchanger face area, m^2
$A_{\min}$	minimum free flow area, m^2
Bi	Biot number ($=\alpha L/\lambda$)
C	thermal capacity, W/K
c_p	specific heat, J/kgK
d	diameter, m
D	thermal diffusivity, m
D_h	hydraulic diameter, m
f	Darcy friction factor, dimensionless
F_f	fin pitch, m
H	height, m
h	specific enthalpy, J/kg
j	Colburn j-factor, dimensionless ($=\text{StPr}^{2/3}$)
L	width, m
m	mass flow rate, kg/s
NTU	number of transfer units, dimensionless
P	depth, m
p	pressure, Pa
Pe	Peclet number ($=GD_h/D$)
Q	heat transfer rate, W
S	pitch, m
T	temperature, K
U	overall heat transfer coefficient, W/m^2 K
UA	thermal conductance, W/K
W	power, W

Greek

α	heat transfer coefficient, W/m^2 K
δ	thickness, m
ρ	density, kg/m^3
λ	thermal conductivity, W/m^2 K
ε	effectiveness, dimensionless
η	efficiency, dimensionless

Subscripts

a	air
cv	control volume
f	fin
i	inlet
lo	longitudinal
o	outlet
r	refrigerant
t	tube
tr	transversal
v	fan

of both the air and the refrigerant streams at the outlet ports, whereas the latter calculates the fan-supplied air flow rate. Both sub-models are described in detail in the following sections.

2.1. Thermal sub-model

The thermal sub-model was divided into two domains namely, air and refrigerant streams. The thermal resistances due to heat conduction through the tube and fin walls were neglected due to the relatively high thermal conductivity of the walls in comparison to the external convection heat transfer (Bi $\sim 10^{-3}$). Both the air and the refrigerant flows were modeled as one-dimensional, steady-state, and purely advective flows due to the relatively low thermal diffusivity of the air in comparison to the face velocity (Pe $\sim 10^3$). Therefore, the air and refrigerant streams were modeled based on the following energy balances applied to the control volume illustrated in Fig. 1:

$$m_r(h_{i,r} - h_{o,r}) + Q_{cv} = 0 \tag{1}$$

$$m_{a,cv}c_{p,a}(t_{i,a} - t_{o,a}) - Q_{cv} = 0 \tag{2}$$

where m_r and $m_{a,cv}$ are the refrigerant and the air mass flow rates through the control volume [kg/s], respectively, h is the refrigerant specific enthalpy [J/kg], and the indices i and o refer to the inlet and outlet ports of the control volume, respectively. The heat transfer rate Q_{cv} was calculated from the concept of heat exchanger effectiveness, as follows:

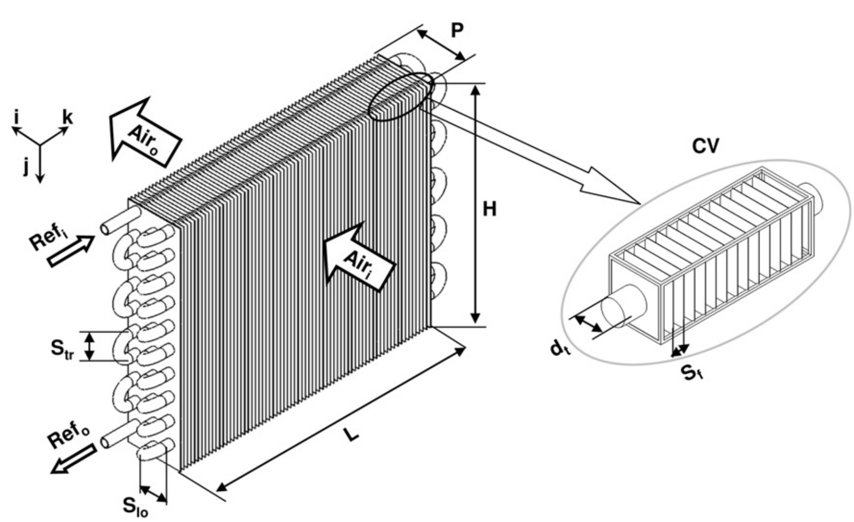

Fig. 1. Schematic representation of the heat exchanger discretization.