International Communications in Heat and Mass Transfer 37 (2010) 98–102

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

CFD modeling of heat transfer of CO_2 at supercritical pressures flowing vertically in porous tubes ☆

Masoud Haghshenas Fard

Department of Chemical Engineering, Isfahan University of Technology, Isfahan, 84156-83111, Iran

ARTICLE INFO

Available online 1 September 2009

Keywords:
CFD
Porous tube
Convection heat transfer
Supercritical pressure

ABSTRACT

Computational fluid dynamics (CFD) tool has been used for investigation of convective heat transfer of CO_2 in two porous tubes. Effects of some important parameters such as pressure, inlet temperature, mass flow rate, wall heat flux and porosity on temperature distribution and local heat transfer coefficients have been studied numerically. Near the supercritical conditions, these parameters are very effective on temperature gradient and local heat transfer coefficients. For example at $p=9.5$ MPa, under the same conditions, the heat transfer coefficient in a tube with particle diameters of 0.1–0.12 mm is about 20–30% higher than when the particle diameter of 0.2–0.28 mm were used. The heat transfer coefficient increases with decreasing pressure and increasing mass flow rate. Also the porosity of the bed has the important role on the heat transfer. The CFD predictions have been compared to the experimental data and showed pretty good agreement.

1. Introduction

Convective heat transfer of supercritical fluid through porous medium has been widely used in many engineering applications such as power engineering, cryogenic and refrigeration processes, water oxidation systems, extraction and separation processes in chemical engineering, transpiration cooling in rocket thruster chambers, heat pumps, and cooling of electric equipments [1–3].

Under the supercritical conditions, small variations in the fluid temperature and pressure can lead to significant changes in the thermo-physical properties, also there will be no phase change with the increase of temperature. In the supercritical region the effect of buoyancy in mixed convection has difference compared to forced convection. The temperature, at which such variations are most noticeable, is called the pseudo critical temperature. In the pseudo critical temperature, the specific heat reaches a maximum value. The pseudo critical temperature for carbon dioxide at 8.5 and 9.5 Mpa are 37.4 and 42.5 °C respectively [4–6].

Fluid flow and heat transfer of fluids through the conduits filled with porous medium have been investigated numerically and experimentally by many researches [7–16].

Jiang et al. [7–10], Yun et al. [11], Yoon et al. [12], Son and Park [13], Pitla et al. [14] Olson and Allen [15], Dang and Hihara [16], experimentally investigated the convective heat transfer of supercritical CO_2 in the circular tube. The results of these experimental data have been compared to existing correlations. Jiang et al. [7–10] presented some experiments to investigate the heat transfer coefficient of CO_2 in a vertical porous tube. In their work, flow resistance in upward and downward flow, and temperature distribution under different conditions have been studied experimentally.

He et al [17], Jiang et al. [18], and Bansod and Jadhav [19], numerically investigated the convective heat transfer in a porous media. In He et al.'s work, computational simulations of experiments on turbulent convection heat transfer of CO_2 at supercritical pressures in a vertical tube have been carried out. It is shown that the buoyancy effect is generally insignificant even when a relatively high heat flux is imposed. Numerical investigation of convection heat transfer of CO_2 at supercritical pressure in a tube at low Reynolds number presented by Jiang et al. [20]. Local heat transfer coefficient and velocity distribution for various heat flux are predicted in their work.

Despite the numerous investigations of heat transfer in a porous media in the past, the CFD methods for investigation of heat transfer of supercritical fluids in the porous tubes were obtained only by few researchers [21,22]. 3-D conjugate heat transfer problem was solved using the CFD software FLUENT 6.1 by Jiang and Lu [21]. In their study, numerical simulation of single-phase fluid flow and convection heat transfer in a porous media is presented. The numerical simulations showed that the temperature distribution at the contact interface is not uniform for the porous media. The two-dimensional finite element method technique has been used in Khanafer et al.'s work [22]. The main objective of their work is examining the effect of the porous sleeve on the buoyancy induced flow motion under steady state condition.

The present study is focused on the CFD analysis of the forced convection of CO_2 at supercritical pressure in a circular tube filled with a saturated porous medium with constant wall heat flux. In addition the undergoing investigation examines the effect of some important

☆ Communicated by W.J. Minkowycz.

E-mail address: Haghshenas@cc.iut.ac.ir.

0735-1933/$ – see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2009.08.004

Nomenclature

B	Body force vector (N/m^3)
C_P	Specific heat capacity (kj/kg·k)
h	Enthalpy (kj/kg)
h_x	Local heat transfer coefficient (W/m^{2}k)
k	Thermal conductivity (W/m·k)
P	Pressure (Pa)
q	Heat transfer rate (W/m^2)
t	Time (s)
T	Temperature (°C)
U	Interstitial velocity vector (m/s)
Greek Symbols	
ε	Porosity
μ	Viscosity (kg/m·s)
ρ	Density (kg/m^3)
τ	Tension tensor (N/m^2)
Subscripts	
w	wall
bf	Bulk fluid

parameters on the heat transfer characteristics. These parameters are wall heat flux, Reynolds number, pressure, inlet temperature, mass flow rate, and particle diameter. The results of CFD simulation are compared to experimental data reported by Jiang et al. [7] and show good agreement.

2. Experimental background

Jiang et al. studied the convection heat transfer of CO_2 in the porous tubes experimentally [7]. Experimental system included a compressed CO_2 container, high pressure pumps, mass flow meter, and pre heater. The test section were porous cylinder tubes with inside an diameter of 4 mm containing particles with diameters of 0.10–0.12 mm (case a) and 0.20–0.28 mm (case b). The test section length was 50 mm. the porosity in the porous tube with articles having diameters 0.10–0.12 mm and 0.20–0.28 mm are 0.45 and 0.4 respectively. The tube walls were heated using electrical resistance wire. The schematic of the test system is shown in Fig. 1.

The parameters measured in the experiments included the wall temperatures, the mass flow rate, the inlet and outlet temperatures, the inlet pressure, the pressure drop across the test section, the electrical resistance, and the heater voltages.

3. CFD modeling

3.1. Governing equations

The CFD approach uses a numerical technique for solving the governing equations for a given flow geometry and boundary conditions. In this paper flow pattern and temperature distribution through the porous tubes were simulated using a commercial CFD package, CFX version 11. The use of CFD reduces the number necessary experiments and gives results, which would hardly be accessible by measurements [23,24].

The general conservation equations describing the single-phase flow through the porous tubes with constant wall heat flux consist of the continuity, momentum and energy equations

– Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \tag{1}$$

– Momentum equation:

$$\frac{\partial}{\partial t}(\rho U) + \nabla \cdot (\rho U U) = -\nabla P + \nabla \tau + B \tag{2}$$

– Energy equation:

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\rho U C_P T) = \nabla \cdot (k \nabla T) \tag{3}$$

where h is enthalpy, T is temperature, k is thermal conductivity, ρ is fluid density, U is the interstitial velocity, τ is the tension tensor, B is the body force, and P is the pressure.

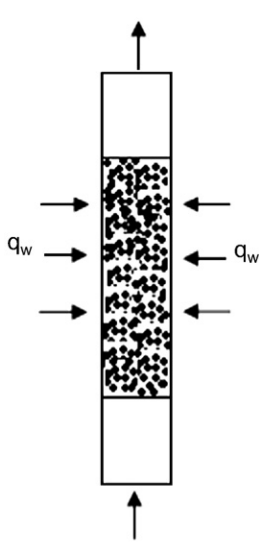

Fig. 1. Schematic of the test system.

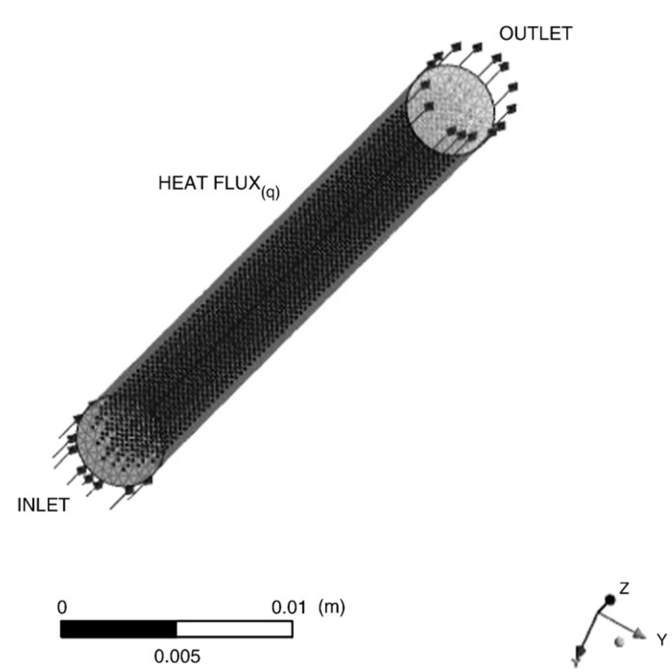

Fig. 2. Computational domain of the problem.