International Journal of Heat and Mass Transfer 112 (2017) 1005–1016

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Drag coefficient and Nusselt number for porous particles under laminar flow conditions

Kay Wittig [a,*], Petr Nikrityuk [b], Andreas Richter [a]

[a] CIC Virtuhcon, Institute of Energy Process Engineering and Chemical Engineering, TU Bergakademie Freiberg, Fuchsmühlenweg 9, 09599 Freiberg, Germany
[b] Department of Chemical and Materials Engineering, Donadeo Innovation Centre for Engineering, The University of Alberta, 9211-116 Str., Edmonton AB T6G 1H9, Canada

ARTICLE INFO

Article history:
Received 9 February 2017
Received in revised form 24 April 2017
Accepted 9 May 2017

Keywords:
Porous particle
Resolved porosity
Array
Immersed boundary
Nusselt number
Drag coefficient

ABSTRACT

This work studies three-dimensional numerical simulations of heat and fluid flow past and through porous particles. Its main objective is to investigate the influence of particle porosity on particle-averaged drag coefficient and surface-averaged Nusselt number numerically. Reynolds numbers considered are in the range of $10 < Re < 250$ and up to 450 for specific porosities. Fluid flow features inside the particles are explored using an immersed boundary method. For this purpose, the particle porosity is modelled using agglomerates of small particles of spherical or cubical shape, respectively. This provides variation of the specific surface area while porosity is kept constant. The simulation results show a significant influence of porosity and Reynolds number on drag coefficient and Nusselt number. For these relationships new formulae are derived and compared to data from the literature.

1. Introduction

An adequate understanding of particle-fluid interaction is of great importance for the modelling of many natural processes (e.g. pollutant transport, particle sedimentation) and industrial applications (e.g. gasification of solid carbonaceous materials). Numerical models used in computer simulations of particulate flows have become important tools in predicting and adjusting the performance of industrial facilities. One which is frequently used, for example, is the Discrete Particle Model by van der Hoef et al. [1] and Zhu et al. [2]. Generally, in those models two phases are considered: a gaseous phase and a disperse phase. Particularly the accurate description of particle movement, or the disperse phase, requires appropriate closures. Closure relations for the particle-fluid interaction and the heat transfer between the particle and fluid are typically represented in the form of the drag coefficient c_D and the Nusselt number Nu, respectively. Most drag coefficient and heat transfer closures have been obtained empirically for ideal spherical and non-porous particles. In reality, however, particles are often porous initially, or become porous, e.g. due to drying or pyrolysis [3,4].

Heat and fluid flow around spheres for a wide range of Reynolds numbers has been extensively studied experimentally, analytically and numerically for a long time [5–10]. In contrast, heat transfer and drag forces acting on a porous spherical body immersed into a fluid have attracted relatively less attention in spite of this subject's importance.

One of the first works devoted to porous particles used semi-analytical models to study the flow around and through an isolated permeable sphere for creeping flow at $Re \approx 0$ [11,12]. These works are of theoretical rather than practical relevance, however. More recently, Bhattacharyya et al. [13] and Yu et al. [14] carried out simplified numerical studies of steady flow around and through a porous cylinder for Reynolds numbers $5 < Re < 40$, Darcy numbers $10^{-6} \leqslant Da \leqslant 1$ and porosities in the range of $0.629 \leqslant \varepsilon \leqslant 0.999$. In both works, single-phase models were utilised where the porosity is modelled implicitly by including Darcy and Forchheimer terms into the Navier-Stokes equation. It was shown numerically that the drag ratio between porous and non-porous cylinders decreases rapidly for $Da < 0.01$. A macroscopic representation of porosity using a permeability approach might be fairly inaccurate with respect to prediction of heat and fluid flow inside a porous particle. Moreover, the prediction of permeability for various porous media is a problem in itself since it depends on the geometry of the pore structure [15,16].

With increase in computational power of modern PCs and development of immersed boundary methods [17,18] it became possible to model porous particles directly resolving at least macro-pores [19]. Due to its straightforward formulation, often only few changes to existing solvers are required for an IB

---

* Corresponding author.
E-mail addresses: wittigkay@gmail.com (K. Wittig), A.Richter@vtc.tu-freiberg.de (A. Richter).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.05.035
0017-9310/

Nomenclature

Roman symbols

A_0	frontal surface of a solid sphere
c_D	drag coefficient
$c_{D,0}$	c_D of a solid sphere
d	particle diameter
d_p	sub-particle diameter
F_{IB}	IB resistance force
F_D	drag force magnitude
K	damping coefficient
K_0	minimum damping coefficient
Ma	Mach number
n_c	number of sub-particle layers
n_p	number of sub-particles
Nu_0	*Nu* of a solid sphere
Nu	Nusselt number
p	pressure
Pr	Prandtl number
Q_{IB}	IB heat source
Re	Reynolds number
Ri	Richardson number
S''	surface ratio
S_0	solid sphere surface
S_{cross}	cross-sectional area
S_{int}	sub-particle coalescence surface
S_p	sub-particle surface
T	temperature
t	time
T^*	dimensionless temperature
T_0	bulk temperature
T_S	particle surface temperature
$\dot{V}$	volume flow
$\dot{V}_0$	volume flow with respect to the inlet
$\vec{v}$	velocity
V_0	solid sphere volume
v_0	bulk velocity magnitude
V_{int}	sub-particle coalescence volume
V_p	sub-particle volume
v_x	x-component of velocity $\vec{v}$
Δx_{min}	minimum cell-width on x-axis

Greek symbols

ε	particle porosity
η	dynamic viscosity
λ	heat conductivity
ρ	fluid density
ϕ	local fluid volume fraction
φ	sphericity

implementation. This led to a wide-spread usage of IB methods for e.g. complex porous media, droplet and particulate flows [20]. Campregher et al. [21] successfully employed an IB method for flows past a solid sphere for *Re* up to 1000. Ilinca and Hétu [22] used in their work an IB method for representing porous media, particularly metal foams, on a grid that does not fit the shape of the immersed object. Their predicted pressure drops for various porous media thicknesses agreed well with data from experiments. Breugem et al. [23] modelled the flow through a random close packing of 9000 beads with two IB methods. In each case they achieved only a 10% lower permeability than measured experimentally by using exactly the same geometry. Finally, an IB method was employed by Nagendra and Tafti [24] for the calculation of fluid flow in more realistic pore spaces. They compare their pressure drop predictions with values computed by use of the Darcy-Forchheimer equation.

Although there are some studies on how a sphere's porosity influences the drag force [25,26], at the same time investigations on the heat transfer at the surface of a porous particle are rare [27]. Thus, main objectives of this work are to evaluate drag coefficient and Nusselt number for different porous particles and the derivation of general relationships for these quantities. Particularly, the flow and heat transfer around a porous sphere is accurately investigated over a wide range of the parameters of porosity ($0.6 < \varepsilon < 0.91$) and *Re* ($10 < Re < 250$), with *Re* ranging up to $Re < 450$ for specific porosity configurations. Based on the performed simulations new formulae are proposed for drag coefficient and Nusselt number of porous particles in dependence on *Re*, particle porosity and surface.

2. Problem description

2.1. Computational setup

First, consider the employed CFD model more precisely. The porous particle is located within a computational domain as illustrated in Fig. 1. Its dimensions $40\,d \times 20\,d \times 20\,d$ are based on the diameter d of the porous particle under investigation such that blockage effects by the particle are negligible. A study on how blockage effects scale with domain size may be found in Richter and Nikrityuk [9]. For the present setup the blockage ratio is 0.2%. Given this domain size, the exact position of the particle is then 10 d short of the inflow and centred in the other two directions. Due to the physical background of this study, the particle size was set to the fixed value $d = 0.788$ mm.

Boundary conditions of the domain are inflow and outflow on the x-axis and symmetrical on the remaining boundaries. A no-slip boundary condition is applied to the solid surface of the porous structure. Boundary temperatures are set to $T_0 = 290$ K for the inflow and to $T_S = 300$ K at the particle surface. All hydrothermal properties of the surrounding fluid are assumed to be constant and correspond to the bulk temperature T_0. Thus, the flow field can be characterised by Reynolds and Prandtl number

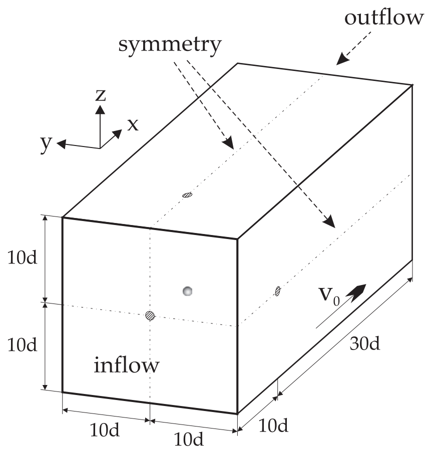

Fig. 1. Computational domain for the flow around a porous particle. The particle is located $10d$ short of the inflow and centred in the directions perpendicular to the flow.