G Model
PARTIC-1082; No. of Pages 10
ARTICLE IN PRESS
Particuology xxx (2018) xxx–xxx
Contents lists available at ScienceDirect
Particuology
journal homepage: www.elsevier.com/locate/partic

Modeling of two-phase particulate flows in a confined jet with a focus on two-way coupling

Babak Yousefi-Lafouraki [a,*], Abas Ramiar [b,*], Ali Akbar Ranjbar [b]

[a] Faculty of Mechanical Engineering, Semnan University, Iran
[b] Faculty of Mechanical Engineering, Babol Noshirvani University of Technology, Iran

ARTICLE INFO

Article history:
Received 16 June 2017
Received in revised form 24 July 2017
Accepted 22 September 2017
Available online xxx

Keywords:
Eulerian–Lagrangian approach
Two-way coupling
Particle tracing
Mixture model

ABSTRACT

A computational fluid dynamics model is used for the simulation of laminar flow of water–Al_2O_3 nanofluid in a confined slot impinging jet. The (steady-state and two-dimensional) Eulerian–Lagrangian model is used considering fluid–particle and particle–wall interactions (i.e., two-way coupling). A collocated grid and the SIMPLE algorithm are used for the coupling of pressure and velocity fields. The deposition model is used to investigate the effect of particle deposition on the impingement surface. Results indicate that the particle trajectory becomes stable farther from the jet with a rising Reynolds number and jet-impingement surface distance ratio. The heat transfer coefficient of the mixture model is higher than that of the Eulerian–Lagrangian model.

Introduction

Nanofluids, in which nanometer-sized particles are diffused in conventional heat transfer fluids, are made to have better thermal conductivity and heat transfer than conventional fluids. They can be used in thermal control systems, with the control of heat transfer being important to technologies of fuel cells and dual fuel–electric vehicles. Heat transfer is also enhanced by the impinging jet. Because of their high momentum transfer rates and high efficiency, impinging jets are widely used in many industrial and engineering applications, such as cooling and heating processes, the drying of textiles, film, paper, and food, the freezing of tissue in cryosurgery and manufacturing, the cooling of gas turbine components and the outer walls of combustors, and the coating and tempering of glass and metal. Recently, investigations on heat and mass transfer with slot jet impingement have attracted attention. Chen, Ma, Qin, and Li (2005) reported that slot jet impingements offer many more beneficial features, such as higher cooling effectiveness, greater uniformity, and more controllability. Slot jets have more applications than circular jets in industry and especially in electronics cooling, because of the larger impingement domain. Over the past 15 years, jet impingement has been used in many applications, such as the cooling of microelectronics and the cooling of insulated-gate bipolar transistors used in hybrid automobiles. Recently, more attention has been devoted to the impinging liquid jet to realize heat transfer rates that are much more difficult to realize using gas jets.

Lee, Song, and Jo (2004) studied the effects of the nozzle diameter on impinging jet heat transfer and fluid flow. They reported that the local Nusselt number increased in the stagnation point region with an increase in the nozzle diameter owing to increasing jet momentum. Chiriac and Ortega (2002) numerically studied steady and unsteady flow and heat transfer for a confined two-dimensional slot jet impinging on an isothermal plate. They reported that in the steady regime, the Nusselt number of the stagnation point increased uniformly with the Reynolds number, and the distribution of heat transfer in the wall jet region was affected by flow separation due to the reentrainment of spent flow back into the jet. Lee, Yoon, and Ha (2008) numerically studied unsteady two-dimensional fluid flow and heat transfer in a confined impinging slot jet using the finite volume method. Their results showed that in the steady region, the time-averaged Nusselt number at the stagnation point increases monotonically with increasing Reynolds number for different height ratios. They also reported that the average skin-friction coefficient in the steady flow decreases with an increasing Reynolds number. Yousefi-Lafouraki, Ramiar, and Ranjbar (2014) numerically investigated the two-dimensional fluid flow and heat transfer of a confined impinging slot jet in a converging channel. Their results showed that two counter-rotating

* Corresponding authors.
E-mail addresses: b.yousefi@semnan.ac.ir (Babak Yousefi-Lafouraki), aramiar@nit.ac.ir (Abas Ramiar).

https://doi.org/10.1016/j.partic.2017.09.008
1674-2001/

Please cite this article in press as: Yousefi-Lafouraki, B., et al. Modeling of two-phase particulate flows in a confined jet with a focus on two-way coupling. *Particuology* (2018), https://doi.org/10.1016/j.partic.2017.09.008

Nomenclature

C_D	Drag coefficient
c_p	Constant pressure specific heat, J/(kg K)
d_p	Diameter of nanoparticles, m
h	Convection heat transfer coefficient, W/(m^2 k)
H	Channel height, m
K	Thermal conductivity, W/(m k)
L	Channel length, m
Nu	Nusselt number
Pr	Prandtl number
Re	Reynolds number
T	Temperature, K
T_b	Bulk temperature, K
V	Velocity, m/s
p	Pressure, Pa
W	Jet width, m
X, Y	Spatial coordinates, m

Greek symbols

μ	Dynamic viscosity, Pa s
ρ	Density, kg/m^3
τ	Shear stress, Pa
α	Converging angle, °
ϕ	Volume fraction of the dispersed phase

Subscripts

ave	Average at the inlet
f	Fluid
c	Continuous phase
jet	Reference (inlet) condition
k	k-th phase
m	Mixture
p	Nanoparticles
w	Wall

vortex structures formed as the jet impinges on the impingement surface and only one stagnation point, where velocity and temperature gradients are high and the maximum Nusselt number occurs at the stagnation point and the highest values correspond to a jet-impingement surface distance ratio of about 5. Nguyen et al. (2009) conducted experiments for Al_2O_3–water nanofluid in a confined and submerged impinging jet on flat, horizontal, and circular heated surfaces for both laminar and turbulent flow regimes. They showed that, depending on the combination of the jet-to-impingement surface distance and particle volume fraction, the use of a nanofluid enhances heat transfer in some cases but has the opposite effect on the convective heat transfer coefficient in other cases. Vaziei and Abouali (2009) numerically studied a circular confined and submerged jet of Al_2O_3–water nanofluid impinging on a horizontal hot plate for both laminar and turbulent flow regimes. Their results showed that the use of Al_2O_3 nanoparticles in laminar jets enhances the heat transfer but for turbulent jets of Al_2O_3–water nanofluid, heat removal is less than that in base-fluid operation.

Single-phase or two-phase flow can be used to simulate the convective heat transfer of nanofluids. In the single-phase model, the nanoparticles and base fluid are assumed to be in thermal equilibrium and they have a no-slip condition. In contrast, slipping between phases because of factors such as gravity, friction between the fluid and solid particles, and Brownian sedimentation and dispersion are considered in the two-phase model. The single-phase model is easier to implement and needs less computational time than the two-phase model. Mirmasoumi and Behzadmehr (2008) and Akbarinia and Laur (2009) numerically studied nanofluid laminar convection in a horizontal tube. In both studies, heat transfer was increased by decreasing the nanoparticle size. Kurowski, Chmiel-Kurowska, and Thulliea (2009) used three models, namely, homogeneous (single-phase), Eulerian–Lagrangian, and mixture models to simulate nanofluid flow inside a minichannel. They reported almost the same results for all models. Fard, Esfahany, and Talaie (2010) studied nanofluid heat transfer inside a tube using both single-phase and two-phase models. Relative to the results of an experimental study on a 0.2% copper–water nanofluid, they reported average relative errors for single-phase and two-phase models of 16% and 8% respectively. Lotfi, Saboohi, and Rashidi (2010) used single-phase, two-phase Eulerian, and mixture models for nanofluid flow inside a tube and reported that, among these methods, the two-phase mixture method is the most accurate.

Particle motion from heat exchanger tubes into a fluidized bed was investigated by Schaflinger et al. (1997). Their numerical results showed that the average particle volume concentration remained nearly constant over a wide range of distances. Pei, Zhang, Ren, Wen, and Wu (2010) studied the hydrodynamics of a gas–solid fluidized bed with two vertical jets using computational fluid dynamics. Their results revealed three flow patterns, consisting of isolated, coalescent, and transitional jets, depending on the nozzle separation distance and jet velocity, which affected the solid circulation pattern. Rouson and Eaton (2001) numerically investigated flow in an inclined channel using one-way coupling. Their results showed that particles tend to accumulate in low-speed streaks near the wall. He, Men, Zhao, Lu, and Ding (2009) numerically studied the convective heat transfer of TiO_2 nanofluids flowing through a straight tube under laminar flow conditions using both single-phase and combined Eulerian–Lagrange models. They concluded that the effects of Brownian motion, thermophoresis, and lifting forces on heat transfer are negligible and found that the heat transfer of nanofluids was enhanced particularly in the entrance region.

This paper numerically investigates the distribution of nanoparticles in a confined slot impinging jet using a two-phase model. Although the mixture model was compared with the Eulerian–Lagrangian model in previous studies (Kurowski et al., 2009; He et al., 2009), the two methods were not considerably different because a straight channel was investigated. Particles do not impact with the wall in a straight channel and are directed to the center of the channel, whereas the impact of particles in the nanofluid with a stagnation point may cause a considerable difference in the impinging jet. The present paper investigates this issue and compares the two models. The effect of the channel inclination angle is also examined.

Model description

Geometrical configuration

Fig. 1(a) shows the impinging jet on a heated wall in the channel. The two-dimensional channel and materials with a steel surface has a length (L) of 0.3 m, height (H) ranging from 0.01 to 0.1 m, and converging angle (α) ranging from 0° to 1.5°, while the jet inlet width (W) is 0.01 m. The H/W ratio ranges from 1 to 10. The considered fluid is a mixture of water and Al_2O_3 nanoparticles with a particle diameter of 100 nm, at different volume fractions of Al_2O_3. The particles are injected into the inlet around the axis of the main jet. The steel elastic properties are obtained from Soltani and Ahmadi (1994).

Boundary conditions

A uniform velocity, V_{jet}, temperature T_{jet} = 293 K, and particle injection are assumed at the channel inlet. The pressure outlet condition is considered at the channel exit section. On the bottom wall,

Please cite this article in press as: Yousefi-Lafouraki, B., et al. Modeling of two-phase particulate flows in a confined jet with a focus on two-way coupling. *Particuology* (2018), https://doi.org/10.1016/j.partic.2017.09.008