Applied Thermal Engineering 113 (2017) 215–228

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Conjugate heat transfer in stratified two-fluid flows with a growing deposit layer

H.Y. Li [a,*], Y.F. Yap [b], J. Lou [a], J.C. Chai [c], Z. Shang [d]

[a] Institute of High Performance Computing, A*STAR, Singapore
[b] Department of Mechanical Engineering, The Petroleum Institute, Abu Dhabi, United Arab Emirates
[c] School of Computing and Engineering, University of Huddersfield, UK
[d] High Performance Research Computing, Division of Research, Texas A&M University, TX 77843, USA

HIGHLIGHTS

- Conjugate heat transfer in two-fluid flows with a growing deposit layer is studied.
- The level-set method is adopted to capture fluid-fluid and fluid-deposit interfaces.
- Deposit layer introduces additional thermal resistance which reduces heat transfer.
- Deposit layer reduces flow area which enhances convection heat transfer.
- Heat transfer depends on strength of thermal resistance and convection enhancement.

ARTICLE INFO

Article history:
Received 30 June 2016
Revised 19 October 2016
Accepted 30 October 2016
Available online 3 November 2016

Keywords:
Conjugate heat transfer
Stratified two-fluid flow
Deposition
Level-set method

ABSTRACT

The article presents a numerical model for moving boundary conjugate heat transfer in stratified two-fluid flows with a growing deposit layer. The model is applicable to other general moving boundary conjugate heat transfer problem in a two-fluid flow environment with deposition occurring simultaneously. The level-set method is adopted to capture the fluid-fluid interface and fluid-deposit interface. The governing equations are solved using a finite volume method. Upon verification of the model, the effects of inlet velocity ratio, Damköhler number and thermal conductivity ratio on the flow, deposition as well as heat transfer are investigated. Generally, Nusselt number on the lower wall (with a growing deposit layer), Nu_{lx} and upper wall, Nu_{ux} show distinct features with the change of these parameters. Nu_{ux} increases with the increase of lower fluid layer (fluid 1) inlet velocity and the thermal conductivity of deposit layer while it decreases with the increase of Damkholer number. Nu_{lx} varies differently in the upstream and the downstream of the channel. A higher lower fluid layer (fluid 1) velocity and a higher thermal conductivity of deposit layer result in a higher Nu_{lx} upstream but a lower Nu_{lx} downstream. However, a higher Damkholer number results in a lower Nu_{lx} upstream and a higher Nu_{lx} downstream.

1. Introduction

A large number of engineering pipe flows involves two immiscible fluids with suspended particles. The prevailing two-fluid flow pattern depends on among others fluids' properties, flow configuration (horizontal, inclined or vertical) and relative flowrate [1]. For example, in a horizontal flow configuration, upon increasing the relative flowrate, the flow pattern progressive changes from bubbly, plug, stratified, wavy, slug to annular flow. Driven either physically or chemically, the suspended particles tend to deposit onto surfaces and form a solid deposit layer. The deposit layer is generally impermeable to flow and introduces extra flow resistance leading to a higher pressure drop. Often in these flows, heat transfer occurs. Heat transfer performance deteriorates because of additional thermal resistance of the deposit layer. Heat is now required to be conducted from the wall across the growing and increasingly thicker deposit layer before transferring to the flowing fluids. Engineering examples include wax deposition in oil-gas [2,3] and oil-water [4,5] flows, asphaltene deposition in oil-water [6] and oil-gas (CO_2) [7] flows, hydrate deposition in water-gas flow [8,9], fouling in two-phase heat exchanger [10] and fouling in flow boiling [11,12].

* Corresponding author.
E-mail address: lih@ihpc.a-star.edu.sg (H.Y. Li).

http://dx.doi.org/10.1016/j.applthermaleng.2016.10.195
1359-4311/

Nomenclature

C	particle concentration (kg/m^3)
c_p	specific heat (J/kg K)
D	diffusion coefficient (m^2/s)
Da	Damkohler number
d	distance (m)
H	height of domain (m)
$H(\phi)$	smoothed heaviside function
k	thermal conductivity (W/m K)
L	length of domain (m)
Nu	Nusselt number
$\hat{n}$	unit normal at the interface
Pe	Peclet number
Pr	Prandlt number
p	pressure (Pa)
$\vec{q}$	deposition flux (kg/m^2 s)
Re	Reynolds number
r_d	reaction rate for deposition (m/s)
S	source term
$S(\phi)$	Sign function
T	temperature (°C)
t	time (s)
$\vec{u}$	velocity vector (m/s)
x,y	Cartesian coordinate (m)

Greek symbols

δ	height of the deposit region (m)
$\delta(\phi)$	Dirac delta function
Γ	interface
θ	dimensionless temperature
ϕ	level set function (m)
φ	component of $\vec{u}_{i,ext}$ (m/s)
ε	interface thickness (m)
μ	dynamic viscosity (kg/m s)
ρ	density (kg/m^3)
Ω	domain of interest

Subscripts

b	bulk
d	deposit
d,ext	extension velocity
f	fluid
lx	local lower wall
ref	reference value
ux	local upper wall
w	wall
x	local
1	fluid 1 region
2	fluid 2 region
$*$	dimensionless

From a modeling point of view, this is a moving boundary conjugate heat transfer problem. There are two boundaries evolving both spatially and temporally, i.e. the fluid-fluid interface and the fluid-deposit front. At these boundaries, various transport processes involving mass, momentum and energy interact with each other in a fully-coupled manner. In particular, coupling of heat transfer in the fluids to that in the deposit layer requires a conjugate approach. The modeling framework generally requires six components to capture (a) fluid-fluid interface, (b) fluid-deposit front; to model (c) fluid transport, (d) particle transport, (e) particle deposition and (f) energy transport. Good prediction of the interaction between transport processes requires accurate determination of the moving boundaries. The fluid-fluid interface can be handled using either a front-tracking approach [13] or a front-capturing approach, e.g. VOF [14] and level-set [15] methods. For fluid-deposit front, apart from VOF and level-set methods, it can also be treated using enthalpy–porosity [16] and total concentration [17] methods. To model particle deposition, i.e. the actual attachment of the particles onto the fluid-deposit front, a critical length coupled with a sticking probability [18,19] or an m-th order deposition reaction [20,21] can be employed. Fluid transport entails prediction of the fluids' velocity and pressure fields. For particle transport, the transient particle distribution is determined using either a Langrangian or Eulerian approach [22]. Energy transport accounts for determining the temperature field. It should be stressed here again that all these six components of the model are fully-coupled together. Modeling then becomes challenging.

Modeling work of such moving boundary conjugate heat transfer problem is scarcely limited in the existing literatures. These existing modeling works will be briefly discussed. To make the problem more tractable, simplifications were often made in the existing modeling works. Therefore, these models do not necessarily have all the six components and may not follow structurally the above framework.

Huang et al. [4] developed a model of wax deposition in a two-dimensional non-isothermal oil-water laminar stratified channel flow. The results presented highlight the importance of incorporating the movement of the oil-water interface for a more accurate deposition prediction, not accounted for in previous studies. For this flow configuration, there exists a priori good geometrical understanding of both the oil-water interface and oil-wax front. The deposition is assumed to be controlled by the particle diffusion into the deposit layer. The flow is modeled as quasi-steady and unidirectional, and thus allowing a simple analytical expression of the velocity field be derived. The interface is then determined such that mass conservation is satisfied. Particle and energy transports are governed only by axial convection and transverse diffusion. The approach suggested serves well for stratified flow but is generally challenging to be extended to other flow configurations with more general interfacial geometries.

Ramirez-Jaramillo et al. [23] proposed a numerical model to simulate asphaltene deposition in a three-phase flow system. These three immiscible phases are oil, gas and water and form a rheological fluid. The flow is determined from flow-pattern specific semi-empirical correlations without tracking or capturing the fluid-fluid interface. Convection heat transfer is considered with empirical correlation used in determining the heat transfer coefficient. Dissolved asphaltene in oil is assumed driven radially by diffusion and precipitates on the wall surface. A thermodynamic model is then utilized to predict this asphaltene precipitation process. In the model, the asphaltene deposit layer formed on the wall is subjected to removal due to shear force. Therefore, the growth of the deposit layer is driven asphaltene precipitation but is retarded by shear removal.

Apte et al. [24] developed a model to investigate paraffin deposition in multiphase flow lines and wellbores. The flow is assumed steady and one-dimensional. For a one-dimensional flow, tracking of the fluid-fluid interface and fluid-deposit front are not required. This greatly simplifies the model. Fluid transport is determined using multiphase mechanistic models for both flow pattern identification and pressure gradient prediction. Regardless of flow pattern, heat transport is modeled by assuming a homogeneous