International Journal of Heat and Mass Transfer 115 (2017) 932–945

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical modeling and simulation of particulate fouling of structured heat transfer surfaces using a multiphase Euler-Lagrange approach

Robert Kasper *, Johann Turnow, Nikolai Kornev

Chair of Modeling and Simulation, Department of Mechanical Engineering and Marine Technology, University of Rostock, Albert-Einstein-Str. 2, 18059 Rostock, Germany

ARTICLE INFO

Article history:
Received 6 June 2017
Received in revised form 20 July 2017
Accepted 24 July 2017

Keywords:
Fouling
Particle-laden flow
LES
Dimples
Vortex structures
Heat transfer

ABSTRACT

A CFD-based multiphase method for the numerical modeling and simulation of particulate fouling of structured heat transfer surfaces is presented and validated using different benchmarks (turbulent particle-laden backward-facing step and channel flow) available in the literature. The proposed procedure is based on a coupling of the Lagrangian-Particle-Tracking (LPT) and Eulerian approach. Therefore, suspended particles are simulated according to their natural behavior by means of LPT as solid spherical particles, whereas the carrier phase is simulated using the Eulerian approach. Large eddy simulations (LES) are performed for fully developed turbulent channel flows at $Re_\tau = 395$ with selected structured surfaces (single square cavity or spherical dimple) considering a depth/diameter ratio of $t/D = 0.26$ and foulant particle mass loading ratios up to $\beta = \dot{m}_p/\dot{m}_f = 2 \times 10^{-3}$ using a dynamic one equation eddy-viscosity turbulence model. These simulations demonstrate the great capabilities of the proposed method and reveal a slightly better fouling performance and thermo-hydraulic efficiency of the spherical dimple, due to the existence of asymmetric vortex structures compared to the square cavity.

1. Introduction

Particulate fouling of heat transfer surfaces due to suspended material (e.g. silt, clay or iron oxide) within the heat exchanger working fluid is still one of the most important problems in heat exchangers and can occur practically on any fluid-solid surface. It has been described as the major unresolved and most challenging problem in heat transfer [1]. Fouling increases the heat transfer resistance and reduces the effectiveness of heat exchangers which causes higher fuel consumption, maintenance costs and costs due to production loss [2]. Despite the fact that particulate fouling reduces the heat transfer and increases the pressure loss, the performance of heat transfer enhancement methods like ribs, fins or dimples is commonly characterized by the thermo-hydraulic efficiency [3–5] or number of transfer units NTU [6], which does not include any information about the fouling behavior. In addition to it, a more or less universal method for the prediction of particulate fouling still does not exist. Existing empirical fouling models (e.g., Kern and Seaton [7] and Taborek et al. [8]) are derived for numerous assumptions and simplifications. Hence, such fouling modeling approaches are unsuitable for a general prediction of particulate fouling and a detailed analysis of fouling mechanisms. An extensive overview about the fundamentals of fouling of heat transfer surfaces and prediction models is given by Bohnet [9], Bott [10] and Müller-Steinhagen [2].

Due to the steadily growing computational resources, the simulation of highly complex processes like particulate fouling using computational fluid dynamics (CFD) becomes more and more important and could be a reliable alternative to expensive experimental measurements. Several numerical studies of heat transfer augmentation methods are available in the literature, which confirms the eligibility of CFD methods for this case of application. For example Casarsa [11] investigated the aerodynamic performance of a fixed rib-roughened internal cooling passage for gas turbine blades. A very efficient way to increase the turbulent heat transfer is the usage of dimpled surfaces, because they allow a significant heat transfer augmentation at low pressure drop penalty, which is approved by detailed experimental studies of Ligrani et al. [12,4] or Mahmood [13]. A substantial numerical study of the flow physics inside a single dimple, including a detailed analysis of the heat transfer enhancement considering the entropy production, has been conducted by Isaev et al. [14], who investigated the influence of the Reynolds number and dimple depth on the turbulent heat transfer and hydraulic loss in a narrow channel using an URANS technique. Well resolved LES of Turnow et al. [15] for a Reynolds number range (based on dimple print diameter) of $Re_D = 20,000$ to 40,000 revealed coherent vortex structures, which

* Corresponding author.
E-mail address: robert.kasper@uni-rostock.de (R. Kasper).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.07.108
0017-9310/

change their orientation in time. Moreover, a three-dimensional proper orthogonal decomposition (POD) analysis on the pressure and velocity fields showed tornado-like spatial POD structures. Additional studies of the thermo-hydraulic efficiency of dimple packages have been performed by Elyyan et al. [16], Lienhart et al. [17] and Turnow et al. [18]. In contrast to several numerical investigations of heat transfer enhancement methods, CFD studies of structured heat transfer surfaces considering particulate fouling are relatively seldom. Latest contributions are from Tong et al. [19], who simulated two-dimensional particle deposition and removal processes on tubes by coupling a multiple-relaxation-time lattice Boltzmann method with a finite volume procedure, and Wang et al. [20] with a parameter study on the fouling characteristic of H-type finned heat exchanger using a RANS approach. Nevertheless, an extensive analysis of the interaction between local flow structures and fouling deposits using transient, large-scale resolving numerical methods such as LES or hybrid RANS-LES (e.g. improved delayed detached-eddy simulation (IDDES) [21]) does not exist at this moment.

The present study is aimed to fill up this lack of knowledge by introducing a new multiphase Eulerian-Lagrangian approach which is suitable for CFD studies of heat transfer enhancement methods under consideration of particulate fouling using large-scale resolving methods. It provides the opportunity to analyze the interaction between local flow structures and different fouling processes in a more comprehensive way. This paper is organized as follows. A detailed description of the proposed method for the simulation of particulate fouling is given in Section 2 with the emphasis on the Lagrangian branch of the method including the procedures of formation and removal of fouling deposits with account for local flow structures. Section 3.1 comprises a profound validation of the general performance of the implemented Lagrangian-Particle-Tracking, particle deposition algorithm and carrier flow simulation using different test cases (e.g. particle-laden backward-facing step flow and fully developed turbulent channel flow including particle-wall deposition) with experimental or numerical data available in the literature. The computational setup for the fouling simulations and a detailed analysis of the flow physics within fully developed turbulent channel flow with a single spherical dimple is given in Sections 3.2 and 3.3. Part 3.4 is dedicated to our first fouling simulations for a plane channel with a spherical dimple or a single square cavity with a comprehensive evaluation of the thermo-hydraulic efficiency considering clean and fouled surfaces. Finally, a section of conclusions completes this paper.

2. Numerical methods

The numerical simulation of particulate fouling on heat transfer surfaces is complex and it mainly consists of the deposition of small suspended particles due to adhesion and sedimentation of larger particles onto horizontal surfaces resulting from gravitational forces. The proposed multiphase method is composed of two different branches, which are closely related to each other. The first one is the Lagrangian branch and describes the physics of the suspended particles or the foulant using the LPT. This branch is mainly responsible for the mass transport of the particles to the heat transfer surfaces, the formation of fouling deposits and also the removal of fouling deposits due to local shear forces. The second one is the Eulerian branch which determines the flow fields of the carrier flow with respect to the fouling deposits.

2.1. Lagrangian branch

The description of isothermal particle motions within a fluid using the Lagrangian-Particle-Tracking (LPT) requires the solution of the following set of ordinary differential equations, to calculate the particle location and the linear as well as the angular particle velocity at any time:

$$\frac{\mathrm{d}\mathbf{x}_p}{\mathrm{d}t} = \mathbf{u}_p, \tag{1}$$

$$m_p \frac{\mathrm{d}\mathbf{u}_p}{\mathrm{d}t} = \sum \mathbf{F}_i, \tag{2}$$

$$I_p \frac{\mathrm{d}\boldsymbol{\omega}_p}{\mathrm{d}t} = \sum \mathbf{T}, \tag{3}$$

where m_p is the particle mass, I_p is the moment of inertia, $\mathbf{F}_i$ includes all forces acting on the particle and $\mathbf{T}$ is the torque acting on the rotating particle due to viscous interaction with the carrier fluid [22]. Newton's second law of motion, Eq. (2), requires the consideration of all relevant forces acting (e.g. drag, gravity and pressure forces) on the particle:

$$m_p \frac{\mathrm{d}\mathbf{u}_p}{\mathrm{d}t} = \sum \mathbf{F}_i = \mathbf{F_D} + \mathbf{F_G} + \mathbf{F_P} + \ldots \tag{4}$$

However, an analytical representation for different forces exists only for small particle Reynolds numbers corresponding to the Stokes regime [23]. Thus, the drag force is expressed more generally in terms of a drag coefficient C_D. The introduction of this drag coefficient allows the calculation of the drag force not only for small particle Reynolds numbers $(Re_p < 0.5)$ where viscous effects are dominating and no separation is observed, but also for the transition region (i.e., $0.5 < Re_p < 1000$) and fully turbulent region or Newton regime (above $Re_p \approx 1000$). The implemented drag model is based on the particle Reynolds number, which is defined as

$$Re_p = \frac{\rho_f D_p |\mathbf{u}_f - \mathbf{u}_p|}{\mu_f}, \tag{5}$$

with the density ρ_f and the dynamic viscosity μ_f of the fluid or continuous phase, the particle diameter D_p and the magnitude of the relative slip velocity $|\mathbf{u}_f - \mathbf{u}_p|$. The drag coefficient is determined using the following drag model based on the correlation proposed by Putnam [24]:

$$C_D = \begin{cases} \frac{24}{Re_p}\left(1 + \frac{1}{6} Re_p^{2/3}\right) & \text{if } Re_p \leqslant 1000 \\ 0.424 & \text{if } Re_p > 1000, \end{cases} \tag{6}$$

which is suitable to higher Reynolds numbers $(Re_p < 1000)$ and ensures the correct limiting behavior within the Newton regime. After determination of the drag coefficient, the basic force representation is used to evaluate the drag force for a spherical particle:

$$\mathbf{F_D} = C_D \frac{\pi D_p^2}{8} \rho_f (\mathbf{u}_f - \mathbf{u}_p) |\mathbf{u}_f - \mathbf{u}_p|. \tag{7}$$

In addition to the drag force, the gravitational and buoyancy force and the pressure gradient force has to be taken into account as well. Within the used LPT, gravitation and buoyancy are computed as follows as one total force

$$\mathbf{F_G} = m_p \mathbf{g} \left(1 - \frac{\rho_f}{\rho_p}\right), \tag{8}$$

where $\mathbf{g}$ is the gravitational acceleration vector. The resultant force due to a local fluid pressure gradient acting on a particle can be defined as

$$\mathbf{F_P} = -\frac{\pi D_p^3}{6} \nabla p. \tag{9}$$

Expressing the pressure gradient ∇p in terms of the differential form of the momentum equation, the force due to a local pressure gradient can be evaluated by