Computers and Fluids 163 (2018) 54–67

Contents lists available at ScienceDirect

Computers and Fluids

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

Modelling ripple morphodynamics driven by colloidal deposition

James N. Hewett*, Mathieu Sellier

Department of Mechanical Engineering, University of Canterbury, Christchurch 8140, New Zealand

ARTICLE INFO

Article history:
Received 11 July 2017
Revised 18 December 2017
Accepted 26 December 2017
Available online 26 December 2017

Keywords:
Evolving boundary
Node shuffle algorithm
Particle deposition
Silica scaling
Dynamic mesh

ABSTRACT

Fluid dynamics between a particle–laden flow and an evolving boundary are found in various contexts. We numerically simulated the morphodynamics of silica particle deposition from flowing water within geothermal heat exchangers using the arbitrary Lagrangian–Eulerian method. The silica particles were of colloidal size, with submicron diameters, which were primarily transported through the water via Brownian motion. First, we validated the Euler–Euler approach for modelling the transport and deposition of these colloidal particles within a fluid by comparing our simulation results with existing experiments of colloidal polystyrene deposition. Then we combined this multiphase model with a dynamic mesh model to track the gradually accumulated silica along the pipe walls of a heat exchanger. Surface roughness was modelled by prescribing sinusoidally–shaped protrusions on the wall boundary. The silica bed height grew quickest at the peaks of the ripples and the spacing between the protrusions remained relatively constant. The rough surface experienced a 20% reduction in silica deposition when compared to a smooth surface. We also discuss the challenges of mesh deforming simulations with an emphasis on the mesh quality as the geometry changes over time.

1. Introduction

Deposition of particles onto a surface through a fluid is a common process found in both nature and industries. For example, sediment is transported by water in the ocean near the seabed and develops wave–formed sand ripples [1]. Deposited material may also detach from the seabed and the net amount of material changes over time, gradually modifying the seabed into ripple shaped structures [2]. This phenomenon is also found in rivers [3] and channels [4]. Experimental [5] and numerical [6] studies have observed migration of these ripples downstream; by analysing sediment dynamics of particles with diameters on the order of 300 μm. Ripple structures have also been observed in biofilms [7], ski moguls [8] and washboard roads [9]. The coupling between the multiphase water and seabed forms an evolving boundary problem. A similar process of particle deposition and resuspension has been studied for industrial pipe flow [10,11]. An application of particle deposition with an evolving surface is 3–D printing [12] where manufacturing of more complex geometries can be achieved when compared to traditional tools.

The context of deposition with an evolving boundary we investigate in this paper is silica scaling which occurs in pipe flow within geothermal power stations. The silica is initially dissolved within water in the underground reservoirs and then, as the mixture reaches the plant equipment at the ground surface, the silica precipitates and forms silica particles [13]. These colloidal (submicron diameter) silica particles are transported through pipe systems within the power plant and may attach to pipe surfaces; gradually accumulating a layer of silica deposit over the period of months or years [14]. This layer of silica adversely affects the performance of heat exchangers. In particular: the pressure loss is increased due to the smaller pipe cross sectional area, and also the heat transfer is reduced as the layer of silica provides an additional thermal resistance.

Various silica scale morphologies have been observed in experiments [15–17] including fibrous deposits, cellular structures and rippled patterns. We numerically simulate the latter deposit structure where rippled structures are aligned normal to the direction of flow (circumferential direction for pipe flow). Numerical simulations and models on silica scaling have previously been explored [18–21]. The focus of this study is the influence of the morphological evolution on the deposition rate by using an evolving boundary model.

One approach for modelling evolving boundary problems is the immersed boundary method [22] where the interface is modelled on a static mesh. An advantage of this approach is not requiring grid transformations [23]; these grid transformations, or remesh-

---

* Corresponding author.
E-mail address: james@hewett.nz (J.N. Hewett).

https://doi.org/10.1016/j.compfluid.2017.12.017

ing, are computationally expensive tasks. However, a refined mesh is important for capturing momentum and concentration boundary layers and therefore a large proportion of the static grid would need to be refined in cases where the interface moves significantly; resulting in a high number of computational cells.

Another approach for modelling evolving boundary problems is tracking the boundary explicitly by modifying fluid properties of the finite volume cells. The block mesh method involves converting fluid cells to solid cells and the interface is defined as the boundary between the fluid and solid zones. This method has been used for modelling fouling in diesel engine exhaust systems [24] and particle deposition on a cylinder in cross flow [25]. However, the boundary between the fluid and solid regions is restricted to the cell faces, resulting in a relatively coarse description of the interface. Furthermore, the resolution of the boundary layer is also restricted by the refinement of the mesh and an excessive number of cells would be required for significant deformations; a similar disadvantage to the immersed boundary method.

A third evolving boundary model is the arbitrary Lagrangian–Eulerian (ALE) method [26–28] which transforms the mesh throughout the simulation to facilitate the boundary deformation. The advantage of this method is that the mesh topology remains constant throughout the simulation and only the individual cells are transformed. The majority of the deformation can be absorbed in the far field region allowing the mesh near the boundaries to remain mostly consistent for the duration of the simulation, and consequently preserving the resolution of the flow gradients in the boundary layer. Fluent, a computational fluid dynamics software, has developed a dynamic mesh model where flows are simulated with a dynamic domain by using deforming boundaries. The boundaries are prescribed with user–defined functions and the mesh interior is dynamically updated at each time step. This model has been validated against experiments with a heart valve [29], with the erosion of a cylinder in cross flow [30] and the melting ice front around a heated cylinder [31]. In this paper, we use the dynamic mesh model, coupled with modelling the silica particle phase, to explore the impact of boundary evolution on accumulation of colloidal silica in pipe flow at the microscale and compare our results with an experimental test rig [32,33].

2. Methods

Two flow configurations were investigated: Poiseuille flow (parallel plates with stationary walls) and Couette flow (parallel plates with the upper wall moving and the lower wall stationary). Both configurations had distinct geometry and particle parameters but had similar physicochemical properties and dimensionless flow dynamics including steady 2–D laminar flow conditions. First, the Poiseuille flow case was used as a validation using existing experimental data to ensure the chosen method of simulating colloidal particle transport in laminar flow was accurate. Polystyrene colloidal particles were modelled within a parallel plate flow cell and their initial deposition rate onto the smooth rigid boundary was compared with experimental [34] and analytical solutions [35]. Second, the accumulation of deposited colloidal silica particles was modelled in turbulent pipe flow where the pipe wall augmented as a function of the silica deposition flux; causing an evolving boundary problem. The pipe wall had a combination of smooth and sinusoidally–shaped protrusions: a single protrusion was simulated and then a series of bumps, forming a surface roughness model. Both configurations were simulated using two phases: one for the fluid, and a second for the dilute suspension of particles. ANSYS Fluent R17.0 was used as the finite volume solver and MATLAB R2016b for the data analysis and visualisation.

Table 1
Mesh convergence (streamwise × wall normal) and validation of particle deposition flux at the centre of the plate, for the Poiseuille flow configuration, compared with the analytical Smoluchowski–Levich approximation.

Mesh	j (#/cm^{2}s)	Error (%)
116 × 10	249.0	−17.3
145 × 30	312.1	3.7
232 × 50	301.9	0.3
464 × 70	300.4	−0.2
Analytic	301.1	

2.1. Geometry and meshing

Dimensionless length scales such as the dimensionless vertical wall distance y^+ are defined as

$$y^+ = \frac{yu^*}{\nu} \tag{1}$$

where y is the vertical distance normal to the surface, u^* the shear velocity and ν the kinematic viscosity. Time t is non–dimensionalised with

$$t^+ = \frac{tu^{*2}}{\nu} \tag{2}$$

and the shear velocity is defined as

$$u^* = \sqrt{\frac{\tau_{w,\text{smooth}}}{\rho}} \tag{3}$$

where $\tau_{w,\text{ smooth}}$ is the wall shear stress in the absence of any sinusoidal protrusions and ρ the fluid density.

2.1.1. Poiseuille flow

The experimental test section [34] had a length of $L = 76$ mm. Half the distance between the parallel plates was $b = 0.3$ mm, and we set an arbitrary depth of 2 mm using one cell in the spanwise direction (across the plates, normal to the flow). We extended the length of the computational domain by 20 mm ($\approx 67b$) in both the upstream and downstream directions in our computational domain to isolate end effects of the inlet and outlet. These developing flow regions were excluded from data analysis and are truncated from the presented results for clarity.

A structured Cartesian grid was generated with uniform cell spacings in the streamwise direction and a bias near the walls in the wall normal direction (between the parallel plates) to capture the near–wall momentum and concentration gradients. An expansion ratio of five was used in the wall normal direction such that the edge lengths of the centre cells were five times that of the near–wall cell widths. Mesh resolutions are listed in Table 1 alongside the results of the mesh convergence analysis.

2.1.2. Couette flow

The experimental test rig [32,33] recirculated fluid containing a dilute suspension of silica nanoparticles through a system of pipes. Silica deposition was observed along a straight pipe section of length $L = 1$ m and diameter $D = 15$ mm. The spatial size of the silica ripples had a height of $h < 50\,\mu$m which is significantly smaller than that of the pipe diameter. The region of interest for this paper is the deposited silica scale which gradually accumulates over time. Therefore a computational subdomain surrounding a ripple was established to avoid unnecessary computations of the far field flow. We assumed that the effect of the evolving ripple had a negligible impact on the bulk flow in the pipe: the blockage ratio is on the order of less than 0.7%.