Comparative Study of the Accuracy of a DNS Solver for Fluid-Gas-Particle Flow Simulation

M. Wu,[1] J.G. Khinast,[1,2] and S. Radl[1]
[1]TU Graz, Institute of Process and Particle Engineering, Inffeldgasse 13/III, Graz
[2]Research Centre Pharmaceutical Engineering GmbH, Inffeldgasse 13/III, Graz
mingqiu.wu@tugraz.at, khinast@tugraz.at, radl@tugraz.at

Abstract

We used our newly developed DNS solver to simulate the gravity driven thin film flow a spherical substrate. This was done to check the accuracy of the solver by comparing the results with the theoretical predictions of Takagi [1].

Introduction

The flow of liquid films is observed in many industries and in nature, e.g., coating applications in the pharmaceutical industry, the spreading of liquid on granules in food processing, the flow of molten iron in welding processes, or the spreading of lava on volcanoes. All these examples have some common features: the fluid is driven by gravity, inertial effects and effects due to a finite wetting speed are often small [1], and the flow is along a rigid solid boundary with complex topology. These flows have been extensively investigated by means of laboratory experiments and theory, revealing key features like an instability that leads to rivulet formation [2]. Unfortunately, theoretical concepts cannot be easily extended to investigate the flow of the liquid into a liquid bridge, which is our long-term objective.

Our work connects to the theoretical and experimental study of Takagi et al. [1], which used lubrication theory to describe the flow of thin films on cylindrical and spherical substrates. In our work we describe a Direct Numerical Simulation (DNS) approach that can be used to study film flow on topologically complex surfaces, and situations where the fluid’s inertia affects the flow. Our DNS approach is based on the immersed boundary (IB) method to model a rigid (but moving) fluid-particle interface, and the volume of fluid approach for modeling a deformable fluid-fluid interface. We use the OpenFOAM software package in combination with a modified version of the CFDEM package [3] to perform efficient DNS in parallel. We check the mass conservation of the liquid droplet after releasing it above a spherical substrate. We compare our simulation results with the theoretical predictions of Takagi [1], and reference DNS using a different treatment of the solid boundary.

Simulation Method

The governing equations for the multi-fluid and fluid-particle system are given as follows:

$$\frac{\partial \vec{u}}{\partial t}+(\vec{u}\cdot\nabla)\vec{u}=-\frac{1}{\rho}\nabla p+\frac{\mu}{\rho}\nabla^2\vec{u}+\vec{f}_s+\vec{f}_p+\vec{g} \tag{1}$$

$$\nabla\cdot\vec{u}=0 \tag{2}$$

$$\frac{\partial \alpha}{\partial t}+(\vec{u}\cdot\nabla)\alpha=0 \tag{3}$$

Where $\vec{u}$ is the fluid velocity, p is the pressure, ρ is the fluid density, μ is the fluid viscosity, $\vec{f}_s$ is the surface tension force, $\vec{f}_p$ is the fluid-particle interaction force, and $\vec{g}$ is the gravitational acceleration. The VOF function $\alpha(0\le\alpha\le 1)$ is an indicator to distinguish between the gas and liquid phase. The local average density and viscosity of the fluid are calculated as follows:

$$\rho=\alpha\rho_l+(1-\alpha)\rho_g \tag{4}$$

$$\mu=\alpha\mu_l+(1-\alpha)\mu_g \tag{5}$$

In our presentation, where ρ_l ρ_g are densities for liquid and gas, μ_l μ_g are viscosities for liquid and gas, respectively. Our DNS approach is based on the immersed boundary method to model a rigid fluid-particle force which corresponds to $\vec{f}_p$ in equation (1), and the volume of fluid approach for modeling a deformable fluid-fluid interaction force corresponded to $\vec{f}_s$ in equation (1).

Results

We present results of a test case in which a small droplet is released on the top of a spherical particle in Figure 1. The droplet starts to spread on the surface of the sphere, and gravitational forces lead to the detachment of droplets from the film. We check the mass conservation problem of the liquid phase value α by decreasing the mesh size of the computational domain gradually. We observe that the loss of liquid phase value α decrease from 9.36% to 5.41% when the mesh size decreases from one-tenth to one twenty-fifth of the particle diameter as shown in table 1. Secondly, we choose the set of parameters of the most accurate case (i.e., case 4 in table 1) to setup the simulation. We then compare simulation results ("IB Solver") with the theoretical prediction ("Theory") of Takagi [1] as shown in Figure 2. The difference between the numerical simulation and the theory is due to the different Bond numbers considered (the Bond number considered in the experiment was 250 time smaller).

Conclusions and Outlook

The DNS predictions of the gas-liquid-particle multiphase flow have been carried out for thin film spreading on a spherical substrate, and the accuracy of the DNS solver has been studied as a function of the grid spacing. We find that extremely fine grids are required to keep the loss in acceptable limits. The simulation results for the liquid spreading rate indicate only a fair agreement with lubrication theory [1], which is valid for the limit of zero Reynolds number. Future work is required to better understand the effect if the fluids' inertia, as well as that of the Bond number Bo_V.

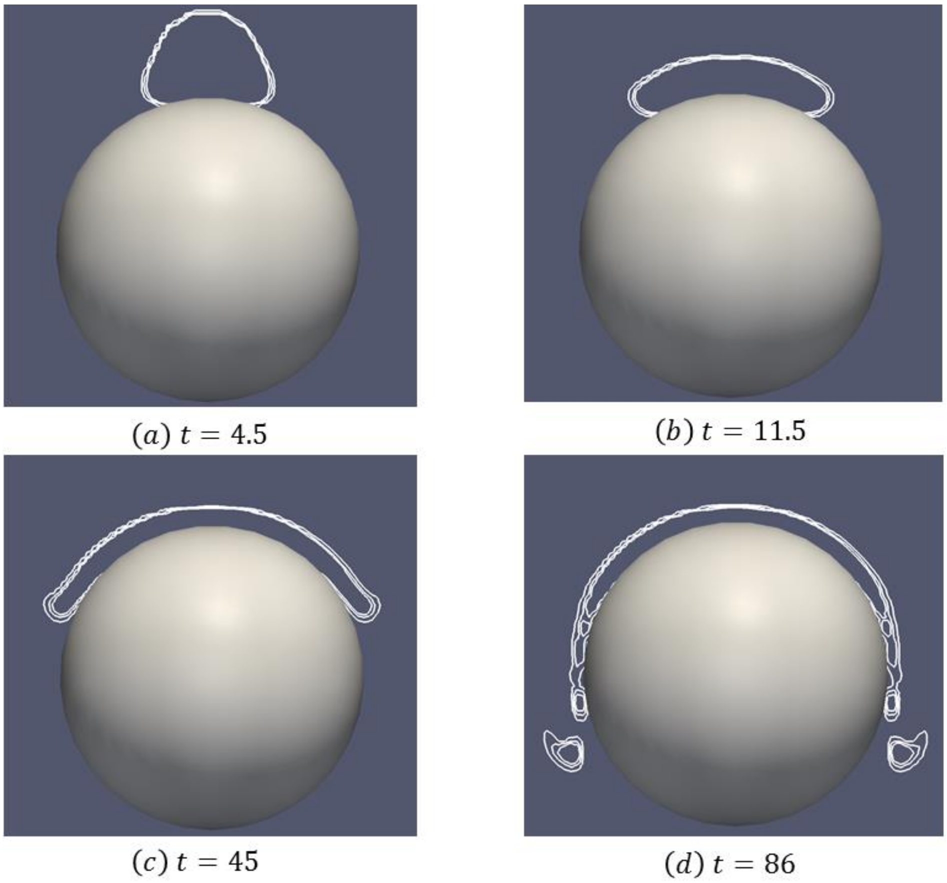

Figure 1: Snapshots from a simulation of liquid spreading on a sphere. (a) t = 4.5 after release, the droplet starts to spread on the sphere. (b) t = 11.5 after the release, the film continues to spread on the surface. (c) t = 45 after release, wave patterns begin to develop at the leading edge of the film. (d) t = 86 after release, droplets form and detach from the film (the reference time scale is $T_{ref} = \nu_l R / gV^{2/3}$*; the dimensionless parameters characterizing the above case are* $Bo_V = (\rho_l g V)/(\gamma R) = 100$, $Re = (\gamma R)/(\rho_l \nu_l^2) = 1$, $\nu_l / \nu_g = 100$, $\rho_l / \rho_g = 10$ *and* $V / R^3 = 0.113$; R, V, γ, ρ, ν, g *are the radius of the sphere, the initial volume of the droplet, the surface tension, the density, the kinematic viscosity, and the gravity, respectively).*

Rel. Domain Size	Rel. Droplet Radius	Grid size Δx	Loss of mass (%)
$4 \times 4 \times 4R$	$0.3\,R$	$d_p / 10$	9.36
		$d_p / 15$	8.81
		$d_p / 20$	8.41
		$d_p / 25$	5.41

Table 1: Mass loss as a function of mesh resolution (d_p is the sphere diameter).

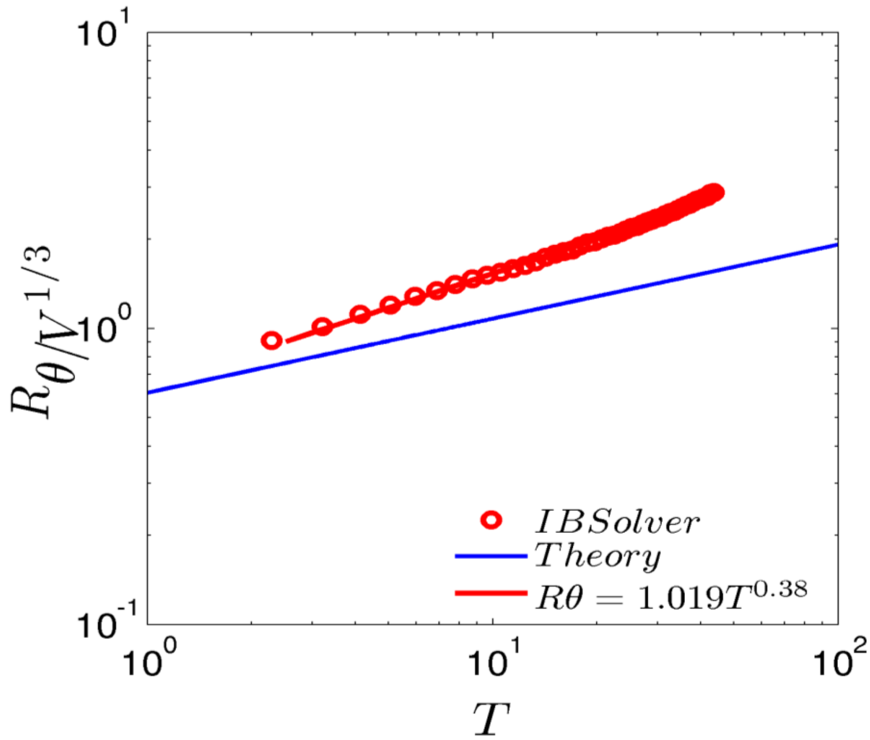

Figure 2: Non-dimensional average flow length of a deformed droplet on top of a sphere as a function of non-dimensional time (dimensionless parameters same as in Figure 1).

Acknowledgement

We acknowledge the funding of the FWF through project P23617 and we gratefully acknowledge support from NAWI Graz by providing access to dcluster.tugraz.at.

References

[1] D. Takagi and Herbert E. Huppert, Flow and instability of thin films on a cylinder and sphere, J.Fluid Mech. 647 (2010) 221-238.

[2] Huppert, H. E, Flow and instability of viscus current down a slope, Nature. 300 (1982) 427-429.

[3] Kloss C., Goniva C. Hager A. Amberger S. Pirker S., Models, algorithms and validation for open-source DEM and CFD-DEM, Progress in Computational Fluid Dynamics, An Int. J. 12,No.2/3 (2012) 140-152