

FLUID–PARTICLE FLOWS IN A DRIVEN CAVITY

P. Kosinski*¹, A. Kosinska¹, and A.C. Hoffmann¹

¹ The University of Bergen, The Department of Physics and Technology, Allegaten 55, N-5007 Bergen, Norway

Key words Rotating fluids, two–phase flows

Subject classification 76U05, 76T10

The research focuses on simulation of flow in a driven cavity containing an incompressible fluid and solid particles. The particle phase was modelled using the Eulerian–Lagrangian (E–L) approach where the solid particles are treated as points moving as a result of the fluid motion in the computational domain. Different cases were simulated: it was shown that increase in particle inertia leads to their faster migration towards the walls of the domain. Moreover the particles influence the fluid flow: the higher initial concentration, the more clear this effect.

© 2005 WILEY-VCH Verlag GmbH & Co. KGaA, Weinheim

1 Introduction

Wall-bounded fluid–particle flows are present in many industrial applications: fluidized beds, pneumatic or slurry transport of dusts, coal combustion, dust explosions, catalytic reactions, piston engines and many others. Of interest is the determination of particle–turbulence interaction or particle–particle interactions and both have been studied in literature mainly by use of the Eulerian–Lagrangian approach (see review by McLaughlin, 1994 [1]).

Rotating flows of viscous fluids have also various applications. The main object of researchers are lid-driven cavity flows, where the fluid is set into motion by a portion of the containing boundary. This type of flows is crucial for analysing fundamental aspects of recirculating fluids: in spite of apparent simple geometry, lid-driven cavity flows may contain high degree of complexity.

Shankar and Deshpande [2] give a comprehensive review on the experimental and numerical studies related to this type of flows. They emphasize the following advantages of driven-cavity flows: (1) results from theory, numerical simulations and experiments can be directly compared; (2) the flow domain is unchanged when the Reynolds number is varied; (3) this class of flows exhibit almost all phenomena that are possible to occur in incompressible flows: eddies, secondary flows, complex 3-D patterns, instabilities, transition to turbulence.

2 Mathematical model

The flow of the continuous phase is modelled using standard Navier-Stokes equations:

$$\nabla \cdot \vec{u} = \vec{0} \quad \rho \left[\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot (\nabla \vec{u}) \right] = -\nabla p + \mu \nabla^2 \vec{u} - \vec{f}_p \quad (1)$$

where: \vec{f}_p expresses the action of the particles on the unit volume of fluid (we are at this stage not yet accounting for the volume taken up by the particles); \vec{u} is the fluid velocity; p is the pressure; ρ is the fluid density, μ is the dynamic viscosity.

The flow of the particles was modelled using the Lagrangian approach where the particles are treated as points, whose motion is the result of the influence of the gas phase [3]:

$$m_j \frac{d\vec{v}_j}{dt} = \vec{f}_{pj} \quad I_j \frac{d\vec{\omega}_j}{dt} = \vec{T}_{pj} \quad (2)$$

* Corresponding author: e-mail: Pawel.Kosinski@ift.uib.no, Phone: +47 555 82817, Fax: +47 555 89440

where for j-th particle: m_j is the mass; \vec{v}_j is the velocity, I_j is the moment of inertia, $\vec{\omega}_j$ is the angular velocity, \vec{T}_{pj} is the torque.

The force \vec{f}_{pj} , which is also used in Eq. 1, describes all forces acting on j-th particle from the surrounding fluid. This is mainly the drag force that may be expressed using the fundamental formula:

$$\vec{f}_{pj} = \vec{f}_{drag} = C_D A_p \rho \frac{|\vec{u} - \vec{v}_j|(\vec{u} - \vec{v}_j)}{2} \quad (3)$$

where: C_D is the coefficient of the drag force (primarily a function of the particle Reynolds number) and A_p is the projected area.

The torque \vec{T}_{pj} can be modelled using the following formula:

$$\vec{T}_{pj} = \frac{\rho}{2} \left(\frac{D}{2} \right)^5 C_R |\vec{\Omega}| \vec{\Omega} \quad (4)$$

where: D is the particle diameter, and $\vec{\Omega} = \nabla \vec{u} / 2 - \vec{\omega}$

The value of the coefficient C_R , used in Eq. 4, is primarily a function of the rotational Reynolds number.

Collisions of particles with walls were modelled using the hard-sphere approach [3]. Collisions between the particles were not modelled since the average distance between the particles was high enough to neglect this issue.

3 Numerical scheme

The numerical scheme for the fluid flow uses finite differences on a structured staggered grid for discretization of derivatives, with Chorin's projection method [4]. The time stepping scheme is explicit. The convective terms were discretized using the Variable-Order Non-Oscillatory Scheme (VONOS) [5], which is of 2nd/3rd order, whereas the pressure laplacian (Poisson equation) was solved using the Successive Overrelaxation method (SOR).

The system of equations, describing the flow of the particles, was solved using the fourth order Runge–Kutta scheme.

4 Results

We analysed a two–phase flow in a driven cavity (see Fig. 1), which describes a flow in a square domain where the upper wall is moving. This leads to a rotation of the fluid in the domain. At the beginning a pure fluid was simulated and the Reynolds number was 1000. When the flow became steady, particles were introduced. We assumed, after our observation, that the steady flow occurred after 500 s.

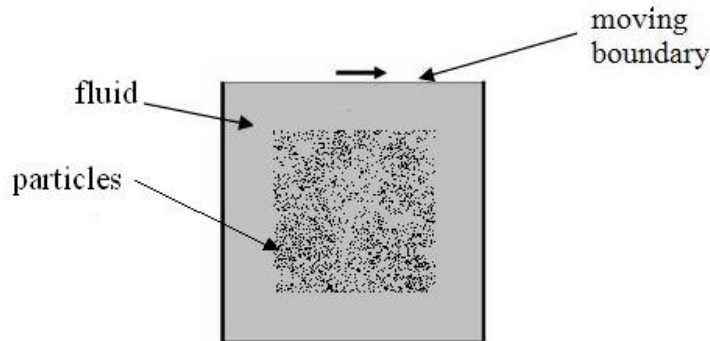


Fig. 1 The description of the problem.

The particles were introduced uniformly with a small displacement in a random direction. They occupied the domain $x \in (0.25, 0.75) \wedge y \in (0.25, 0.75)$.

In this paper we limit ourselves to presentation of the results of the following four cases:

- Stokes number: 0.02778, 2.778
- Relative distance between the particles: $L/D = 50$ or 25

In the above: L is the real distance between the particles

The Stokes number is defined as:

$$St = \frac{\tau_v}{U} = \frac{\rho_p d^2}{18\mu} \quad (5)$$

where: U is the velocity of the moving boundary, τ_v is the particle momentum response time, ρ_p is particle density

Figure 2 shows snapshots of particle position for two points in time: 2.5 and 5 s. The Stokes number was 0.02778. This figure shows how the rotating fluid puts the particles into motion and how the particles are pushed against the walls of the cavity. When the Stokes number increases to 2.778 the process is similar (see Fig. 3). Nevertheless, due to higher inertia the particles encounter the wall earlier than for the lower value of Stokes number. They are namely put into motion by the fluid, but do not react promptly to the rotation of the fluid.

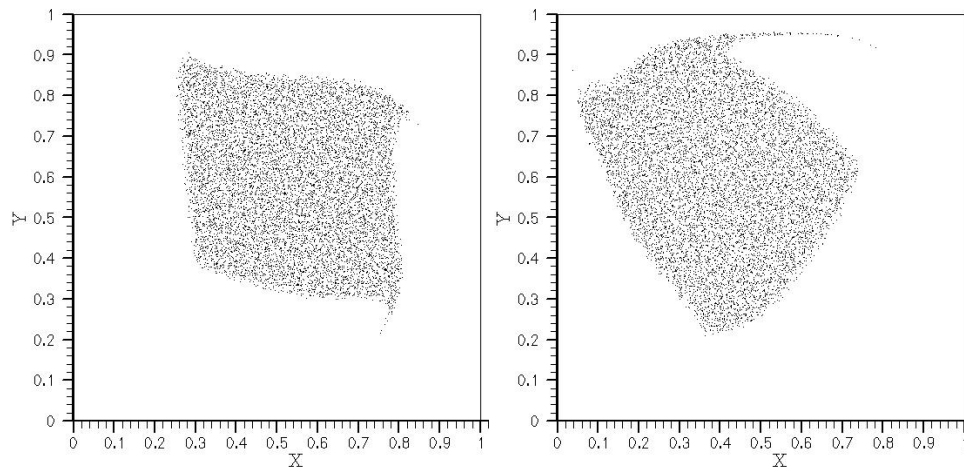


Fig. 2 Snapshots of particle position for two points in time: (a) 2.5 s, (b) 5.0 s. $St = 0.02778$

In the model, so-called two-way coupling was assumed, which means that particles may influence the flow of fluid. This is shown in Fig. 4 as the history of x-component of fluid velocity monitored at point (0.25, 0.25). Before the particles were introduced (time 500 on the graph) the velocity was nearly constant. After the particles have been distributed in the cavity, the velocity started to vary. This is more intense for higher Stokes numbers and for smaller distance between the particles (higher concentrations).

5 Concluding remarks

The paper focuses on simulation of two-phase flow in a driven cavity for the Reynolds number equal to 1000. Different cases were considered, where the Stokes number and the initial distance between the particles varied. It was shown how the particle cloud behaves in the system and simultaneously how the particles influence the flow of the fluid. In this paper we showed that when the Stokes number increases and this distance decreases, the change in fluid velocity is more clear. A similar issue has been also addressed before by many other researchers so our next step is to describe this in a quantitative way in the near future.

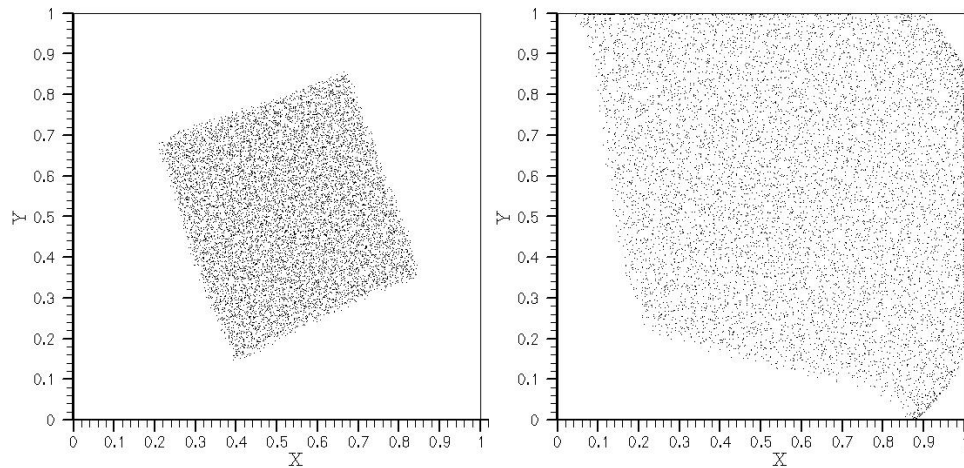


Fig. 3 Snapshots of particle position for two points in time: (a) 2.5 s, (b) 5.0 s. $St = 2.778$

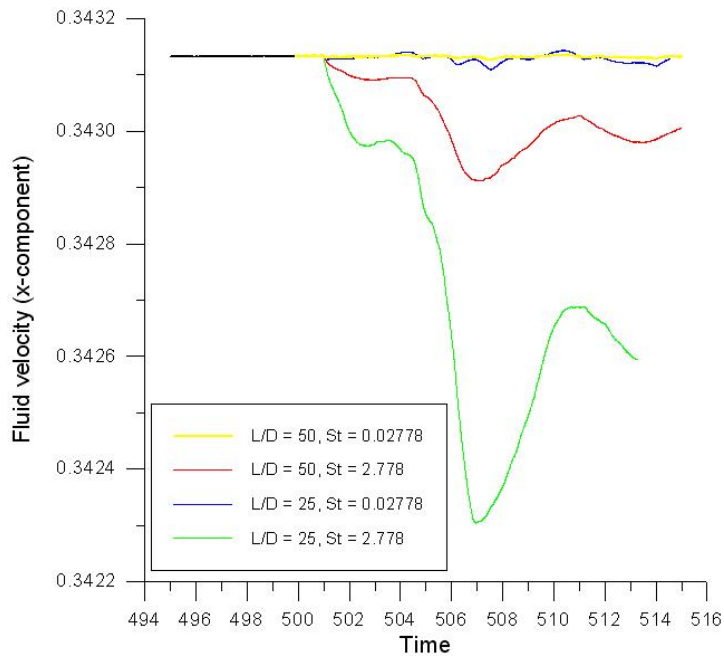


Fig. 4 History of x-component of fluid velocity at point (0.25, 0.75) after the particles have been introduced, for different values of the Stokes number and initial distance between the particles

Acknowledgements This research was financed by the Research Council of Norway and Norsk Hydro.

References

- [1] J.B. McLaughlin, *Int. J. Multiphas. Flow* **20**, 211–232 (1994)
- [2] Shankar, P., Deshpande, M., *Annu. Rev. Fluid. Mech.* **32**, 93–136 (2000).
- [3] Crowe, C., Sommerfeld, M., Tsuji, Y., *Multiphase flows with droplets and particles* (CRC Press LLC, 1998)
- [4] Griebel, M., Dornseifer, T., Neunhoeffler, T., *Numerical simulation in fluid dynamics. A practical introduction.* (Society for Industrial and Applied Mathematics, 1997)
- [5] Varonos A., Bergeles G., *Int. J. Numer. Meth. Fl.* **26**, 1–16 (1998)