European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2006
P. Wesseling, E. Oñate and J. Périaux (Eds)

THE PHYSALIS METHOD FOR PARTICLE SIMULATIONS

Johanna J. Bluemink[*1], Aurore Naso[*] and Andrea Prosperetti[†,*]

[*] University of Twente,
Faculty of Science and Technology and
J.M. Burgers Center for Fluid Dynamics,
P.O. Box 217, 7500 AE Enschede, The Netherlands
e-mail: j.j.bluemink@tnw.utwente.nl
web page: http://pof.tnw.utwente.nl/

[†] Johns Hopkins University,
Department of Mechanical Engineering,
Baltimore MD 21218
e-mail: prosperetti@jhu.edu

Key words: Navier-Stokes Flows, No-slip Condition, Sphere(s), Finite Difference Method, Rigid Body Motion, Spherical Harmonics

Abstract. *This paper illustrates the* Physalis *method designed for the simulation of Navier-Stokes flows with suspended rigid spheres. The method exploits the no-slip condition to linearize the flow about a rigid body motion in the immediate vicinity of each particle. In this way an analytic solution valid within a mesh length near the particle surface becomes available, while the solution at a greater distance is obtained by a standard finite-difference projection method. It should be stressed that no parameterization of the hydrodynamic force or torque on the particles is introduced: these quantities are calculated from first principles.*

The analytical basis and numerical implementation of the method used in this work are described in several other publications (Takagi et al. 2003; Zhang & Prosperetti 2003, 2005) and we give here only a very brief description.

The fundamental – as well as obvious – observation is that, due to the no-slip condition, at the surface of each particle the fluid velocity is exactly equal to a rigid body velocity field. By continuity, the deviation of the actual fluid velocity from this rigid-body velocity field in the immediate neighborhood of the particle will be small and its square can therefore be neglected. On the basis of this remark, it is possible to introduce an auxiliary divergenceless velocity field $\tilde{\mathbf{u}}$ which vanishes at the particle surface and to an excellent approximation satisfies the Stokes equations in its neighborhood. It should be stressed that this procedure is **not** a linearization of the Navier-Stokes equation about 0, as the

[1]Work supported by FOM and STW

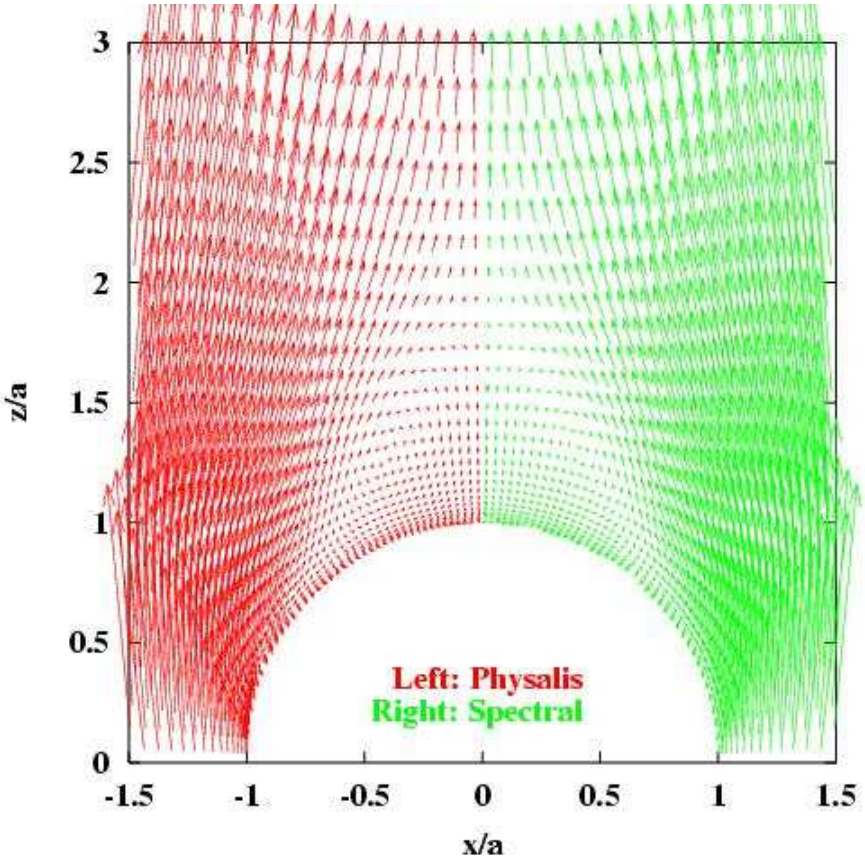

Figure 1: The flow field in the near wake of a sphere in a uniform incident flow at $Re = 50$ as predicted by the present method (left) and by the spectral calculation of Bagchi & Balachandar (2001, 2002).

original Stokes equation, but about the rigid-body motion of the particles. Furthermore, the approximation is used only up to a distance of the order of one mesh size from the particle surface and, therefore, the magnitude of the error can be controlled by controlling the discretization. The general solution of the Stokes equations can be expressed analytically in terms of undetermined coefficients.

At the same time as we consider these local spectral representations of the solution valid near each particle, we construct by finite-differences a solution of the full Navier-Stokes equations in the interstitial fluid. The advantage of this two-fold representation of the solution is that, rather than imposing the boundary conditions on the finite-difference solution at the particle surface, we demand that the finite-difference solution match the local solution at the points of a suitable *cage* of nodes surrounding the particle. The nodes are part of the finite-difference grid and, therefore, the geometric complexity that arises from the mismatch between the regular grid and the particle boundaries is completely avoided. The finite-difference grid is constructed to cover the entire domain, irrespective of the presence of the bodies. A standard staggered grid arrangement is used with pressure

at cell centers, velocities at the midpoint of cell sides, and vorticity components at the midpoints of the cell edges. Each particle is surrounded by a cage of cells with the respective grid nodes for velocity, pressure, and vorticity.

We use an iteration procedure which can be summarized as follows. Suppose that a provisional estimate of the flow fields, such as the flow fields at the previous time step, is available. Then:

1. For each particle, we match the provisional pressure and vorticity fields to their local analytical representations at the respective cage nodes to generate a linear system for the undetermined coefficients of the Stokes solution.

2. By using the values of the coefficients determined at the previous step, from the analytic formulae we calculate the velocity components at the velocity points of the cage.

3. We solve the full Navier-Stokes equations on the finite-difference grid imposing this velocity field as boundary condition at the velocity nodes of the particle cages.

4. We calculate the corresponding pressure and vorticity, return to step 1, and repeat until convergence.

5. After convergence of the iterations, the position, velocity, and angular velocity of each particle are updated by a second-order discretization of the pertinent equations.

The results of a typical validation test are shown in Fig. 1 which depicts a comparison of the velocity field behind a sphere at a Reynolds number of 50 as calculated by the present method and by a spectral method (Bagchi & Balachandar 2001).

The method has been designed to simulate many suspended particles. Figure 2 shows the fluid kinetic energy in the course of a simulation of 1,024 falling particles at a single-particle Reynolds number of about 10.

Details of the method are described in the following papers, listed in reverse chronological order:

REFERENCES

[1] Z. Zhang, L. Botto and A. Prosperetti. Microstructural effects in a fully-resolved simulation of 1,024 sedimenting spheres, *Proceedings of the IUTAM Symposium on Computational Multiphase Flow*, Kluwer, to appear.

[2] Z. Zhang and A. Prosperetti. A second-order method for three-dimensional particle flow simulations, *J. Comput. Phys*, **210**, 292-324, (2005).

[3] S. Takagi, H.N. Oğuz, Z. Zhang, Z and A. Prosperetti. *PHYSALIS*: A new method for particle simulation. Part II: Two-dimensional Navier-Stokes flow around cylinders, *J. Comput. Phys*, **187**, 371-390, (2003).

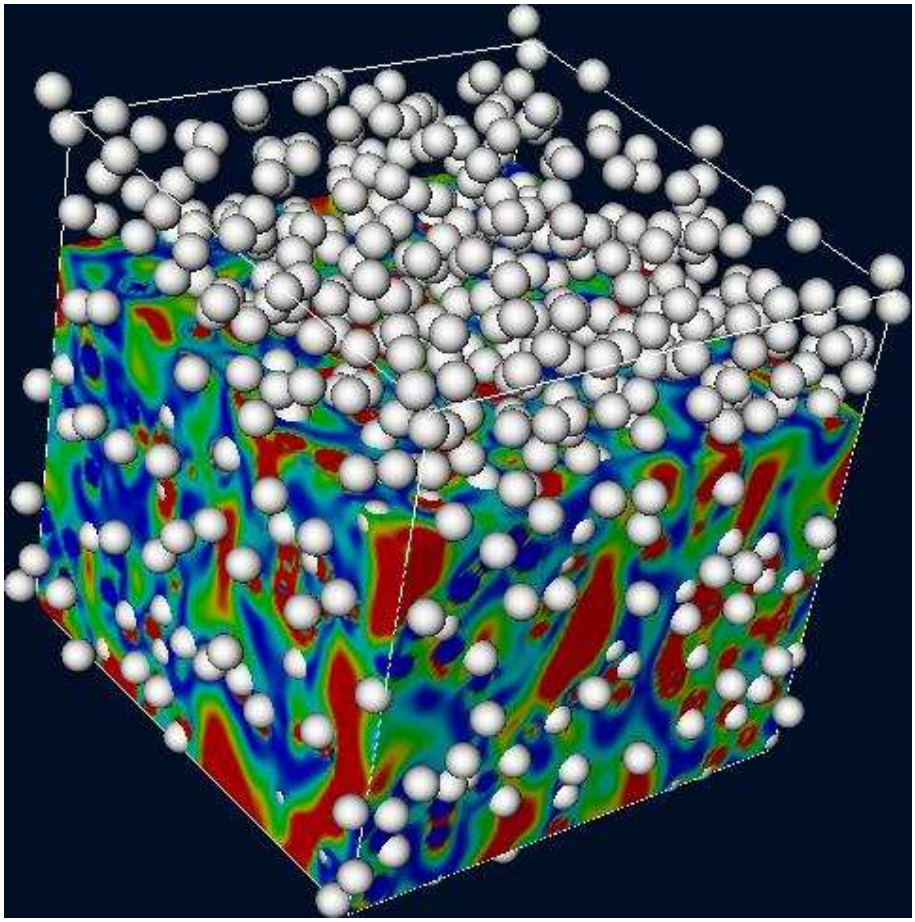

Figure 2: Snapshot of the particle arrangement and fluid kinetic energy at nondimensional time $tw_t/a = 16$ (w_t is the terminal velocity of a single particle and a the particle radius).

[4] Z. Zhang, and A. Prosperetti. A method for particle simulations, *J. Appl. Mech.*, **70**, 64-74, (2003).

[5] A. Prosperetti and H.N. Oğuz. *PHYSALIS*: A New $o(N)$ Method for the Numerical Simulation of Disperse Systems. Part I: Potential Flow of Spheres, *J. Comput. Phys* **167**, 196-216, (2001).