A NUMERICAL STUDY OF LOW REYNOLDS NUMBER STRATIFIED FLOW PAST A SPHERE

Dany De Cecchis, Carlos Torres , José Castillo , Germán Larrazábal

[Contact: dececchi@sciences.sdsu.edu]

Introduction

We model the flow past a sphere in a viscous, incompressible low Reynolds number stratified fluid. The flow has uniform velocity and linear stratification. The governing set of equations, Navier-Stokes, describing the above flow were numerically solved by direct integration using boundary-fitted coordinates and a Pressure Correction Method with finite-difference approximations. In order to solve the pressure linear system with high accuracy we use the BiCGstab method. Numerical experiments are performed for different flow conditions by varying the Reynolds number. Results are presented for density and velocity fields. Simulated sphere drag is also compared to the classical drag curve cited in the literature

Governing Equations

We consider a sphere moving either horizontally or vertically at a constant speed in a uniformly stratified fluid and the density is diffusive. The dimensionless governing equations are

$$\frac{D\mathbf{u}}{Dt} = -\nabla p - \frac{1}{F^2}\rho\hat{\mathbf{z}} + \frac{2}{Re}\nabla^2\mathbf{u},$$
$$\frac{D\rho}{Dt} = u - 1 + \frac{2}{Pe}\nabla^2\rho,$$
$$\nabla\cdot\mathbf{u} = 0,$$

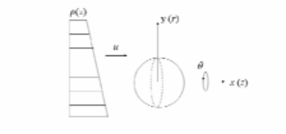


Figure 1: A schematic of horizontal problem

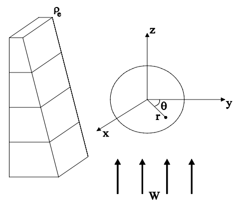


Figure 2: A schematic of vertical problem

where t is the time, $\mathbf{u} = (u, v, w)$ is the velocity, p is the pressure, ρ the density, $\hat{\mathbf{z}}$ the unit vertical vector. The dimensionless numbers, Reynolds (Re), Froude (F), Schmidt (Sc) and Pléclet (Pe) are defined as

$$Re \equiv \frac{2Ua}{\nu},\quad F \equiv \frac{U}{Na},\quad Sc \equiv \frac{\nu}{\kappa},\quad Pe \equiv Re\ Sc,$$

here, U is the velocity, N the Brunt-Väisälä frequency, a the radius of the sphere, μ the viscosity, ν= μ/ρ is the kinematic viscosity, κ the diffusivity coefficient. For each kind of problem is needed to impose suitable boundary conditions, they are:

Horizontal

$\mathbf{u} = 0$ on the sphere,

$\mathbf{u} = (1,0,0),\ \rho = -z$ on the upstream boundary,

$\frac{\partial\mathbf{u}}{\partial x} = \frac{\partial\rho}{\partial x} = 0$ on the outer boundary

Vertical

$\mathbf{u} = 0,\ \nabla\rho\cdot\mathbf{n} = 0$ on the sphere,

$\mathbf{u} = \hat{\mathbf{z}}$ on the upstream (lower) boundary,

$\nabla\mathbf{u} = 0$ on the downstream (upper) boundary,

$\rho = 0$ on the outer boundary

In whatever case as upstream is located far enough from the sphere, these conditions are fixed to its unperturbed value and time independent. The total drag coefficient $C_D = C_f + C_p$ (friction and pressure), where

$$C_p = \frac{1}{\frac{1}{2}\rho_0 U^2\pi a^2}\int_S(-p\delta_{i,k})n_k\,dS,\quad C_f = \frac{1}{\frac{1}{2}\rho_0 U^2\pi a^2}\int_S\mu\left(\frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i}\right)n_k\,dS,$$

with i denotes the mean flow direction (vertically upward), n_k is the component of the unit vector normal to the sphere surface and dS is the area unit of the surface integral.

Computational Scheme

We use a PCM (Pressure Correction Method), then is necessary define a "pressure diagnostic Poisson equation"

$$\nabla^2 p = -\frac{1}{F^2}\nabla\cdot(\rho\hat{\mathbf{z}}) - \nabla\cdot[(\mathbf{u}\cdot\nabla)\mathbf{u}] + \frac{2}{Re}\nabla^2 D - \frac{\partial D}{\partial t},\quad D = \nabla\cdot\mathbf{u}$$

with suitable boundary conditions. We use a linear implicit Euler method to split the time development, and all spatial derivative are approximated by the second order 11-stencil finite different scheme obtaining the linear systems

$$\mathbf{M}_p p^n = f_p^n,\quad \mathbf{M}_\mathbf{u}^n\mathbf{u}^{n+1} = f_\mathbf{u}^n,\quad \mathbf{M}_\rho^n\rho^{n+1} = f_\rho^n.$$

Where the pressure system does not change, but must be solve with high accuracy. A BiCGstab method preconditioned with a dual threshold incomplete LU factorization is used. Each velocity component and density systems change in time, and they are solved using SOR applied in the mesh.

References

[1]. H. HANAZAKI **A numerical study of three-dimensional stratified flow past a sphere**. *Journal of Fluid Mechanics*, 192:393--419, 1988.

[2] LARRAZÁBAL, G., C. R. TORRES and J. E. CASTILLO. **An efficient and robust algorithm for 2D stratified fluid flow calculations**. J. *Appl. Num. Math.* 47 493--502, 2003.

[3]. C. R. TORRES, H. HANAZAKI, J. OCHOA, J. CASTILLO, and M. VAN WOERT. **Flow past a sphere moving vertically in a stratified diffusive fluid.** *J. Fluid Mech.*, 417:211--236, 2000.

[4]. C. R. TORRES, J. OCHOA, J. E. CASTILLO, and H. HANAZAKI. **Numerical simulation of flow past a sphere in vertical motion within a stratified fluid.** *J. Comp. and Appl. Math.*, 103(1):67--76, 1999.

[5]. H. A. VAN DER VORST. **Bi-CGSTAB: A fast and smoothly converging variant of Bi-CG for the solution of non-symmetric linear systems**. *SIAM J. Sci. Stat. Comput.*, 12:631--644, 1992.

[6]. Y. SAAD. ILUT: **A dual threshold incomplete LU factorization.** *Numer. Linear Algebra and Appl.*, 1:387--402, 1994.

Numerical Scheme

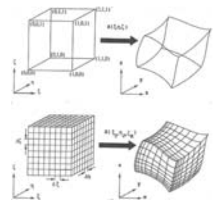
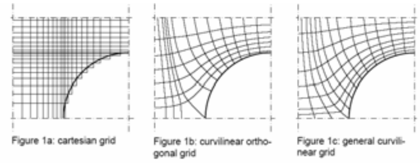


Figure 3. Mapping from physical to computational domain, using general curvilinear coordinates



From H. Knaus, J. Maier et al. Advantages of Applying Boundary-Fitted Grids to the Simulation of Pulverised Coal-Fired Utility Boilers with Mixed Staging Burners. (e-mail: knaus@IVD.Uni-Stuttgart.DE)

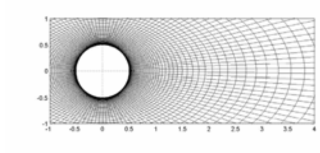


Figure 4. A detail of the grid near the sphere in physical space

The discretization of the domain use a general curvilinear coordinate, generating a no orthogonal boundary fitted grid. The external boundary of the grid is elliptic with a size of 20 sphere diameters in the vertical and 40 sphere diameters in the horizontal. The grid consisted of 131×91×48 (ξ× η× ζ) mesh points in (*z, r, θ) space*

Numerical Results and Simulations

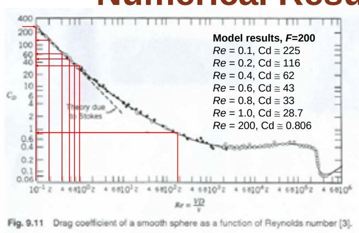
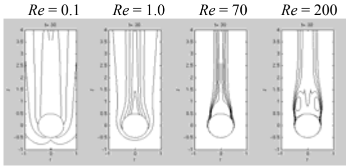


Fig. 9.11 Drag coefficient of a smooth sphere as a function of Reynolds number [3].

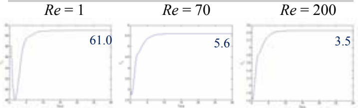
The numerical results were compared with experimental results, when F=200 (Fluid no stratified). The numerical result agree well in a wide range of Re.

$F = 200$

$Re = 0.1$ $Re = 1.0$ $Re = 70$ $Re = 200$

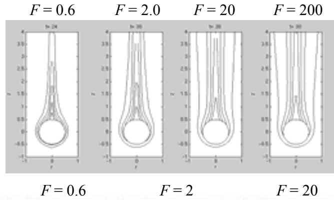


$Re = 1$ $Re = 70$ $Re = 200$

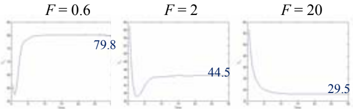


$Re = 1$

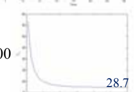
$F = 0.6$ $F = 2.0$ $F = 20$ $F = 200$



$F = 0.6$ $F = 2$ $F = 20$



$F = 200$



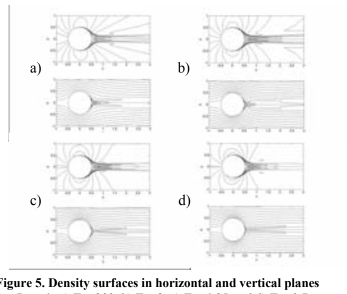


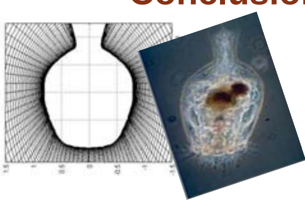
Figure 5. Density surfaces in horizontal and vertical planes for *Re = 1: a) F = 200, b) F = 2, c) F = 1.25 and d) F = 0.7. Density contours are drawn for ρ – z = ±0.2, ±0.4, ±0.6, etc.*

Different Profiles of Density Perturbation and values of Drag Coefficient are shown whether vertical and horizontal fluid.

Re =1	C_D
F = 20	28.05
F = 2	29.09
F = 1.25	30.18
F = 0.7	32.00

Table 1. Drag Coefficient for different values of Froude.

Conclusions & Future Works



- ✓ Greater stratification higher drag value.
- ✓ More detailed study when density plot does not show separation of fluid .
- ✓ Use another techniques to time splitting, to achieve more robustness.
- ✓ More efficient implementation of code.
- ✓ Simulate with some microorganism shape.