J. Math. Study
doi: 10.4208/jms.v52n3.19.04

Vol. **52**, No. 3, pp. 277-298
September 2019

A Semi-Langrangian Vortex Penalization Method for 3D Incompressible Flows

Chlo Mimeau[1,*], Iraj Mortazavi[1] and Georges-Henri Cottet[2]

[1] *Laboratoire M2N Modélisation, Mathématique et Numérique,*
Conservatoire National des Arts et Métiers, 292 Rue St Martin, 75003 Paris, France.
[2] *Laboratoire Jean Kuntzm ann, Univ. Grenoble Alpes and CNRS,*
700 Avenue Centrale, 38041 Grenoble, France.

Received December 27, 2018; Accepted April 20, 2019

Abstract. A remeshed Vortex method is proposed in this work to simulate three-dimensional incompressible flows. The convection equation is solved on particles, using a Vortex method, which are then remeshed on a Cartesian underlying grid. The other differential operators involved in the governing incompressible Navier-Stokes equations are discretized on the grid, through finite differences method or in spectral space. In the present work, the redistribution of the particles on the Cartesian mesh is performed using a directional splitting, allowing to save significant computational efforts especially in the case of 3D flows. A coupling of this semi-Lagrangian method with an immersed boundary method, namely the Brinkman penalization technique, is proposed in this paper in order to efficiently take into account the presence of solid and porous obstacles in the fluid flow and then to perform passive flow control using porous medium. This method, which combines the robustness of particle methods and the flexibility of penalization method, is validated and exploited in the context of different flow physics.

AMS subject classifications: 65M22, 35Q30, 76S05

Key words: Semi-Lagrangian method, remeshed Vortex method, penalization approach, bluff body flows, DNS.

1 Introduction

In recent years, improvements in computing enabled a large enhancement of numerical simulations related to Computational Fluid Dynamics (CFD). These simulations allow to predict the physical behavior of fluid flows. In the present study we particularly focus on

---

[*]Corresponding author. *Email addresses:* `chloe.mimeau@cnam.fr` (C. Mimeau), `iraj.mortazavi@cnam.fr` (I. Mortazavi), `georges-henri.cottet@univ-grenoble-alpes.fr` (G.-H. Cottet)

incompressible Newtonian fluid flows past obstacles, which have been and are still subject to many investigations due to their close relation to aerodynamic efficiency. In that sense, numerical simulations constitute a real economic issue when applied to engineering problems and are considered as interesting and reliable alternatives to wind-tunnel experimentations.

Among the numerous numerical approaches used in CFD, Lagrangian methods, also called particle methods, occupy an important place thanks to their intuitive and natural description of the flow. Indeed, in Lagrangian approaches, the physical quantities involved in the simulated problem are discretized onto a set of particles evolving spatially in the domain according to the problem dynamics. The particles are therefore characterized by their position in the computational domain and the value of the physical quantity they are carrying. Vortex methods [1] belong to this class of Lagrangian approaches and will constitute the key point of the present work. In vortex methods, the particles discretize the Navier-Stokes equations in their velocity ($\mathbf{u}$) - vorticity ($\boldsymbol{\omega}$) formulation, closed with a Poisson equation allowing to recover the velocity of the fluid flow from the vorticity field. This formulation allows to directly point to the essence of vorticity dynamics in incompressible flows, which is characterized by advection and diffusion as well as stretching, which denotes the change of orientation. Another important feature of Lagrangian vortex methods lies in their low numerical diffusion [1,2] and their stability.

However, vortex methods exhibit difficulties inherent to particle methods, mostly related to the treatment of the boundary conditions and the distortion of the particle distribution, which manifests itself by the clustering or spreading of the flow elements in high strain regions, thus implying the loss of convergence of the method. As demontrated first in [3] and later in [4], the convergence of Lagrangian vortex methods relies on the particles overlapping : if the vorticity carried by a particle is spacially distributed on a blob, that is to say on a disk of finite radius ε, then the convergence of vortex methods implies a strict relation between the particle spacing h and the blob radius ε, more precisely one must ensure $h=\mathcal{O}(\varepsilon)$. The remeshing technique [5,6] may be considered as one of the most efficient and popular method to bypass the inherent problem of particle distribution distortion. It consists in periodically redistributing the particles onto an underlying Cartesian grid, while the momentum equation is solved in a Lagrangian framework, in order to ensure their overlapping and thus the convergence of the solution. These remeshed vortex methods involve a Lagrangian framework for the advection and stretching problems while handling also a fixed Cartesian grid. The presence of this grid facilitates the prescription of the no-slip boundary conditions as well as the modeling of the diffusive term and the resolution of the Poisson equation, using Eulerian schemes (e.g finite-differences, spectral methods, ...), and ensures the particle overlapping condition. In a computational point of view, the Cartesian grid also provides a simple and efficient framework in terms of implementation and parallelization.

In this work, we present a remeshed vortex method coupled with an immersed boundary technique in order to account for the presence of obstacles in the flow and to model the boundary conditions. The immersed boundary method chosen here is the Brinkman