Available online at www.sciencedirect.com
ScienceDirect
Comput. Methods Appl. Mech. Engrg. 302 (2016) 147–169
Computer methods in applied mechanics and engineering
www.elsevier.com/locate/cma

Simulating 2D viscous flow around geometries with vertices through the Diffused Vortex Hydrodynamics method

E. Rossi[a,c], A. Colagrossi[a,*], D. Durante[a], G. Graziani[b]

[a] *CNR-INSEAN, Marine Technology Research Institute, Rome, Italy*
[b] *DIMA, Department of Mechanical and Aerospace Engineering, Sapienza University of Rome, Italy*
[c] *LHEEA Lab. (ECN / CNRS), École Centrale Nantes, Nantes, France*

Received 13 July 2015; received in revised form 6 January 2016; accepted 8 January 2016
Available online 19 January 2016

Abstract

The Diffused Vortex Hydrodynamics (DVH) is a recent numerical model born as an evolution of the classical vortex particle methods, where an improvement of the solution quality has been achieved through a regularization of the particles spatial distribution during the vorticity diffusion process. The DVH method is a pure meshless method which adopts a body-fitted approach to enforce no-slip boundary condition on solid surfaces. In the present work it is exploited to perform an accurate analysis of the vorticity field generated by the incompressible flow around bodies with geometrical singularities. In common academic or technological applications, such singularities are quite frequent (*e.g.* trailing edges of wing profiles or of propeller blades) and the classical Euler mesh-based methods can suffer in modeling these geometries. In the recent years a wide literature was produced in order to describe enhanced numerical methods that could overcome such challenging problems. The potentialities of the DVH approach to problems where bodies with geometric singularities are involved are deeply discussed and some examples are finally offered.

Keywords: Vorticity dynamics; Vortex method; Particle methods; Meshless methods; Diffused Vortex Hydrodynamics; Flow around geometrical singularities

1. Introduction

In the last decades, the accurate analysis of the vorticity field generated by a body in motion played an important role in technology applications: aircraft design, helicopter, bridges or skyscrapers are clear examples. Moreover, it proved to be fundamental and strategic for new renewable energy devices. The research on wind turbines farms, in fact, covers a large literature on this topic and a new and interesting challenge is represented by the bladeless evolution of them which exploits the tower oscillations induced by the shed vorticity (see e.g. Villareal [1]).

* Corresponding author. Tel.: +39 06 50 299 343; fax: +39 06 50 70 619.
E-mail address: andrea.colagrossi@cnr.it (A. Colagrossi).

http://dx.doi.org/10.1016/j.cma.2016.01.006
0045-7825/

The numerical simulation of the vorticity field generated by complex bodies represents a non trivial task even at the present days: the algorithms based on the solution of the Navier–Stokes equations in terms of fields (e.g. Finite Differences, Finite Volumes of Finite Elements) or particles (e.g. Smooth Particle Hydrodynamics) are not directly affected by the vorticity calculation as a primary variable, but its evaluation is usually performed in the post processing with an unavoidable accuracy degradation. Moreover, the classical approaches to the description of the flow field around a body generally suffer when geometric singularities are involved, as it may be typical in the naval hydrodynamic environment. The presence of bodies with sharp surfaces frequently forces to exploiting unstructured meshes, that are simpler to generate but less accurate in the convergence control with respect to the structured ones. However, the latter requires a certain regularity of the domains mapped, so a wide literature on numerical techniques, developed in order to overcome these issues, has been produced in the last decades (see for general description of overlapping grids technique Petersson [2] or for naval applications Muscari et al. [3]). In the paper of Marrone et al. [4] the δ-SPH approach is addressed in order to study violent water impacts. There, the flow past a sharp-edged obstacle shows a particular effort in dealing with the geometric singularity. To tackle this problem, a ghost-fluid approach was formulated in order to ensure the enforcement of the correct boundary conditions.

Differently from the outlined approaches, the vortex particle methods allow to study the vorticity evolution in a proper way, being the vorticity the primary variable of the problem. In the present paper the Diffused Vortex Hydrodynamics (DVH), presented in Rossi et al. [5,6] is exploited for the modeling of 2D incompressible viscous flow around bodies with geometrical singularities. The DVH method has been validated in Rossi et al. [5] where the evolution of vorticity distribution in free space is considered. In Rossi et al. [6] the flow at moderate and high Reynolds numbers past smooth bluff bodies of various shapes has been studied and the results were compared with those present in the literature.

The use of a vortex method for incompressible flows brings several advantages (see e.g. Cottet and Koumoutsakos [7], Koumoutsakos and Leonard [8], Chorin [9]):

(i) the pressure field is no longer a direct unknown of the problem when the Navier–Stokes equations are written in vorticity formalism,
(ii) the continuity equation is automatically satisfied,
(iii) the vorticity formulation allows to discretize only the rotational region of the flow (self-adaptivity),
(iv) high accuracy on the evaluation of the velocity field (because it is obtained through a spatial integration),
(v) possibility to simulate flows at high Reynolds numbers,
(vi) the boundary conditions at infinity are automatically satisfied, therefore large spatial domains are not required to correctly enforce them,
(vii) finally, advantage comes from the Lagrangian nature of the method that reduces the numerical dissipation present in mesh-based approach coming from the nonlinear term of the Navier–Stokes equations (see e.g. Cottet and Koumoutsakos [7]).

The DVH method exploits the operator splitting algorithm introduced by Chorin [9]. The single time step is divided in two sub-steps: one advective and one diffusive. In the advective sub-step, the velocity field is evaluated through a fast multipole method (FMM, see e.g. Graziani and Landrini [10]) while, in the diffusive sub-step,the diffusion of the vortices is performed using the deterministic algorithm described by Benson et al. [11]; the latter being based on a superposition of elementary solutions of the heat equation. If a body surface exists within the fluid domain, the no slip boundary condition is enforced through the generation of a vortex sheet, as described in Chorin [9].

In the diffusive step each vortex particle spreads its circulation on a Regular Point Distributions (RPD). At the end of this step, a new set of vortices is generated on the nodes of the RPD overwriting the previous one. This procedure avoids the excessive clustering or rarefaction of the vortices without using any remeshing (with interpolation procedure). To solve the diffusion near a smooth solid boundary, an homogeneous Neumann condition for the vorticity field is used, together with a flat plate approximation of the solid contour itself. However, the latter approximation is no more valid for the vortices close to geometrical singularities, no matter the level of the spatial discretization adopted.

In this work, the DVH method is used to simulate flows around bodies with non-smooth boundaries. The presence of geometrical singularities requires the modification of the diffusion algorithm described in Rossi et al. [6] with the introduction of a suitable visibility mask algorithm. This method is then tested around bodies of increasing complexities.