©SNAK, 2015

Int. J. Nav. Archit. Ocean Eng. (2015) 7:1034~1043
http://dx.doi.org/10.1515/ijnaoe-2015-0072
pISSN: 2092-6782, eISSN: 2092-6790

Further validation of the hybrid particle-mesh method for vortex shedding flow simulations

Seung-Jae Lee[1], Jun-Hyeok Lee[2] and Jung-Chun Suh[1,2]

[1]*Research Institute of Marine Systems Engineering, Seoul National University, Seoul, Korea*
[2]*Department of Naval Architecture and Ocean Engineering, Seoul National University, Seoul, Korea*

Received 20 March 2015; Revised 23 August 2015; Accepted 26 August 2015

ABSTRACT: *This is the continuation of a numerical study on vortex shedding from a blunt trailing-edge of a hydrofoil. In our previous work (Lee et al., 2015), numerical schemes for efficient computations were successfully implemented; i.e. multiple domains, the approximation of domain boundary conditions using cubic spline functions, and particle-based domain decomposition for better load balancing. In this study, numerical results through a hybrid particle-mesh method which adopts the Vortex-In-Cell (VIC) method and the Brinkman penalization model are further rigorously validated through comparison to experimental data at the Reynolds number of* 2×10^6*. The effects of changes in numerical parameters are also explored herein. We find that the present numerical method enables us to reasonably simulate vortex shedding phenomenon, as well as turbulent wakes of a hydrofoil.*

KEY WORDS: Vortex shedding; Hydrofoil; Vortex-in-cell (VIC); Penalization; Large eddy simulation (LES).

INTRODUCTION

Propeller singing is a critical vibration phenomenon generated by the interaction between a Kármán vortex shedding mechanism from the trailing-edge of a blade and its natural frequency (Fischer, 2008). The singing phenomenon has been interpreted as the self-excited oscillation simulated by a closed loop. That is to say, Kármán vortex shedding mechanism is a self-excited system which can continue to shed vortices periodically without any periodic stimulation from outside (Shioiri, 1965). Vortex-Induced Vibration (VIV), which is the most common problem in fluid-structure interaction, is of particular interest to practical ocean engineers (for example, Chen and Kim, 2010; Kim and Rheem, 2009). In such a flow, the volume of fluid with significant vorticity magnitude is typically a small fraction of the total flow volume. The flow can be represented in a more compact form by vorticity than velocity (Stock et al., 2008). An unsteady turbulent wake behind a hydrofoil is therefore considered as a good prototype to assess the viability of numerical approaches based on velocity-vorticity formulation.

Vortex methods are essentially a grid-free approach, and flows are basically represented as a set of vortex particles tracked in a Lagrangian manner. The importance of vortex methods lies in the possibility of analyzing directly the vorticity which plays a great role in fluid dynamics. Chen and Marshall (1999) present a summary of the benefits of vortex methods compared to more traditional approaches based on the velocity-pressure formulation on a fixed grid. Vortex methods remain as an interesting

Corresponding author: *Seung-Jae Lee*, e-mail: *hul94@snu.ac.kr*

alternative to finite difference or finite volume approaches. However, high computational cost of vortex methods is a leading cause limiting their widespread use. This is the reason why vortex methods have been developed toward an efficient algorithm that combines Lagrangian and Eulerian schemes. In recent years, a significant reduction of the computational time has been achieved through the combination of the Vortex-In-Cell (VIC) method pioneered by Christiansen (1973) and the Brinkman penalization approach (Angot et al., 1999), as well as the practical use of parallel computer systems (also refer to Coquerelle and Cottet, 2008; Ossmani and Poncet, 2010; Rasmussen et al., 2011; Beaugendre et al., 2011; Lee et al., 2014). There are both a fixed (Eulerian) mesh and moving (Lagrangian) particles. In a so-called hybrid particle-mesh method, field quantities such as velocity and vorticity (or circulation) are typically solved on a temporary uniform mesh and then interpolated back to particles. Particles move freely in *n*-dimensional space as individuals according to their velocity vectors during a unit time interval. The VIC method requires much less computing time compared with the Fast Multipole Method (FMM), as well as direct summation methods (Cocle et al., 2008), and the penalization technique is possible to replace an usual vorticity creation algorithm for enforcing a no-slip condition at a solid body. It is therefore noted that the VIC method combined with Brinkman penalization technique offers an efficient way to simulate incompressible viscous flows and enables one to track a larger number of vortices. Another good feature of the hybrid particle-mesh method is that an immersed boundary approach greatly simplifies creation of meshes by decoupling solid boundaries and computational meshes.

In this paper, we present numerical investigations of turbulent flows behind a NACA 0009 hydrofoil with a truncated trailing-edge to assess the capability of our numerical methodology. The current work also aims at analyzing the effects of changes in key parameters for numerical simulations.

GOVERNING EQUATIONS AND NUMERICAL METHODS

For a two-dimensional flow parallel to *xy*-plane, the velocity-vorticity form of the Navier-Stokes equations can be expressed in a Lagrangian frame as

$$\frac{D\mathbf{x}}{Dt} = \mathbf{u} \quad and \quad \frac{D\omega_z}{Dt} = \nu\nabla^2\omega_z \tag{1}$$

where ν is the kinematic viscosity and ω_z is the scalar plane component of vorticity vector. N discrete Lagrangian fluid particles are advanced with the corresponding local velocity $\mathbf{u}$, and their strength is gradually diffused due to viscous effects. The pressure term in the Navier-Stokes equations is completely decoupled in this form and not part of the solution algorithm.

In the hybrid particle-mesh method, the right-hand sides of Eq. (1) are efficiently evaluated on a uniform Cartesian mesh using finite difference schemes. The vorticity ω_z is first interpolated onto a uniform mesh by

$$\omega_z(\mathbf{x}_m) = \frac{\varepsilon^2}{h^2}\sum_p^N \omega_z(\mathbf{x}_p) W(\frac{\mathbf{x}_m - \mathbf{x}_p}{h}) \tag{2}$$

Here h is the mesh spacing and W is the interpolation kernel. Each particle is characterized by its position $\mathbf{x}_p$ and its strength Γ. A circulation $\Gamma = \int \omega dS \approx \omega\varepsilon^2$ where ε is the size of the fluid particle. The subscript m and p denote mesh and particle quantities, respectively. In the current work, the third order interpolation kernel M_4' (Monaghan, 1985) is used. A single particle contributes to the nearest 16 nodes through the M_4' kernel, and the total vorticity at each node is obtained by summing the vorticity contributions of all the vortex particles.

VIC method

The stream-function on the uniform mesh is computed through the FFT-based Poisson solver based on an open-source library called Fastest Fourier Transform in the West (FFTW) (Frigo and Johnson, 2005). Thereafter, rotational velocities on the nodes of the grid are computed from the definition $u_x = \partial\psi_z / \partial y$ and $u_y = -\partial\psi_z / \partial x$ through the fourth order central finite