J. Wind Eng. Ind. Aerodyn. 151 (2016) 122–136

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

Flow reproduction using Vortex Particle Methods for simulating wake buffeting response of bluff structures

Samir Chawdhury *, Guido Morgenthal

Institute of Modelling and Simulation of Structures, Bauhaus University Weimar, Marienstraße 13, 99423 Weimar, Germany

ARTICLE INFO

Article history:
Received 1 October 2015
Received in revised form
29 December 2015
Accepted 2 February 2016
Available online 24 February 2016

Keywords:
Vortex Particle Method
Bluff body aerodynamics
Wind engineering
Vortex shedding
Wake
Computational efficiency
Wake buffeting

ABSTRACT

The paper presents a novel extension of the two-dimensional Vortex Particle Method which allows complex transient flow features computed by an original simulation to be recreated for use as inflow conditions in other simulations. This is facilitated by recording velocity time signals of the original simulation and computing time traces of vortex particles to be seeded into the secondary simulation near its upstream domain boundary. The proposed Flow Reproduction Method (FRM) thus allows us to re-create the flow field, without the need to simulate the underlying physics responsible for the flow features. A natural field of application is the re-creation of wakes from flows past bluff bodies of arbitrarily complex geometry, the resolution of which is computationally expensive. The recording is performed on a sampling system of the velocity field. Reproduction of the sampled simulation is then performed by inserting vortex particles in defined positions and time intervals into the secondary simulation. This proceeds in a smaller domain with the advantage of significantly reduced computational cost. In the simulations presented here, wake flows of upstream cylinders are reproduced. Convergence studies are performed to validate the FRM. The quality of flow reproduction is assessed and quantified. The computational efficiency of the reproduction simulation is enhanced additionally by using different adaptive numerical techniques. The method is then applied to fluid-structure interaction simulations of a wake buffeting problem. Good agreement is found between the dynamic response quantities when comparing original simulations and those with reproduced flows, whereby the latter are performed up to five times faster.

1. Introduction

Modeling of flows past bluff bodies is a challenging topic and numerous numerical methods have been developed for application in this domain. Experimental studies such as Nakamura and Nakashima (1986), Nakamura et al. (1996), and Saha et al. (2000) are always considered as standard procedures, however the advantages that make the numerical methods increasingly popular are their ability to predict the full-scale aerodynamic behavior and clear visualization of interesting flow phenomena around bluff bodies (Williamson, 1996; Norberg, 1993). The numerical approaches such as the Vortex Particle Method (VPM) have gained significant interest in recent years, particularly in the direction of bluff body aerodynamics (Irwin, 2008; Koumoutsakos and Leonard, 1995; Liu and Kopp, 2011; Ploumhans and Winckelmans, 2000; Slaouti and Stansby, 1992). The VPM is principally based on the particle discretization of the vorticity field in a Lagrangian form of the governing Navier–Stokes equations. It has been found to be computationally very efficient for solving two-dimensional (2-D) fluid-structure interaction problems over the grid-based methods, e.g. Finite Volume or Finite Element methods, and no body-fitted computational mesh is required at moving boundaries. The highly promising VPM have gained considerable attention in research and practice, e.g. they have been used for the analysis of complex aerodynamic problems of several bridge sections (Ge and Xiang, 2008; Hejlesen et al., 2015; Larsen and Walther, 1997, 1998; McRobie et al., 2013; Morgenthal, 2005; Prendergast, 2007; Rasmussen et al., 2010; Taylor and Vezza, 2009; Walther and Larsen, 1997; Zhou et al., 2002).

This paper presents a novel extension of 2-D VPM for performing reproduction of simulation of complex flow fields that are generated from flows past bluff bodies of arbitrary geometry. The purpose of flow reproduction is to model specific simulated wake fields only by inserting particles into a secondary VPM simulation. The Flow Reproduction Method (FRM) primarily requires velocity sampling from downstream of a chosen simulation of flows around bluff bodies. The velocity sampling is performed till the end of the simulation using a series of predefined square

* Corresponding author. Tel.: +49 3643 584421.
E-mail address: samir.chawdhury@uni-weimar.de (S. Chawdhury).

http://dx.doi.org/10.1016/j.jweia.2016.02.003

sampling-cells across the flow direction. The particle-based VPM necessitates the transformation of sampled fluctuating velocity components of each square sampling-cells into vortex particles. Finally, the approximated vortex particles are inserted into the reproduction simulation to replicate the sampled flow fields. Importantly, the particles are released into the free stream flow of the new or reproduction simulation exactly at the same rate of the velocity sampling. Here, the approximated vortex particles are the only computational elements that carry the information of the fluctuations in flows coming from the upstream bodies.

The concept of seeding vortex particles for modeling of 2-D unsteady wind was initially presented in Prendergast (2007) and Prendergast and McRobie (2006). The vortex particles were approximated from the externally modeled statistically defined unsteady wind before seeding them into the VPM simulation. The mentioned concept of modeling the oncoming turbulence by seeding vortex particles was furthermore applied in Hejlesen et al. (2015) and Rasmussen et al. (2010) for the simulation and estimation of the aerodynamic admittance in bridge aerodynamics. The FRM generalizes the concept to arbitrary velocity signals and applies it to recorded flow features. Preliminary results of the implementation of FRM were presented in Chawdhury and Morgenthal (2015) and Ibrahim et al. (2014).

The simulation of wake flows coming from an upstream square section at Reynolds number *Re* of 500 is used here as a reference case. Convergence studies are performed for the validation of the Flow Reproduction Method with respect to the number of seeded particles. The size of the square sampling-cells and the rate of velocity sampling determine the number of particles for seeding. The quality of flow reproduction is assessed quantitatively by comparing the flow profiles using far downstream flow monitoring sampling points. The instantaneous flow fields are compared including the errors induced in the flow reproduction. Different adaptive numerical techniques, e.g. the particle remeshing presented in Morgenthal and Walther (2007), are employed in FRM which allow the reproduction simulations to be performed computationally faster than original simulation.

The vibration of structures induced by oncoming wakes from upstream bodies presents a critical phenomenon which may govern their design, e.g. in fatigue-related wake galloping cases of stay cables of cable-supported bridges (Xu, 2013). In this study, as an application field of FRM, the method is used for performing wake buffeting analyses of a downstream rectangular section under original and reproduced wakes coming from an upstream square. The differences in maximum and root mean square displacements of the system are compared for validation of the scheme in wake buffeting analysis. The efficiency of using computationally effective reproduced wakes in the wake buffeting analysis is studied by comparing simulation run-times. This way the FRM is presented as an effective method for modeling inflow wakes from upstream bodies to perform faster wake buffeting analysis. The method may be very advantageous for cases where particular inflow conditions shall be re-used multiple times, e.g. for system optimization or sensitivity analyses. As a representative study, the influences of different amounts of system damping on the vibration of the downstream rectangular section under original and reproduced wakes from upstream square are studied.

The investigations within this study have been carried out using a Computational Fluid Dynamics (CFD) solver named VXflow (Morgenthal, 2002), which is fundamentally based on VPM. A lot of improvements over the classic VPM make this implementation effective for modeling of flows past complex structural assemblies (Morgenthal and Walther, 2007). The code has been validated on a range of fundamental studies and has successfully been used to study the aerodynamic behaviors of several bridges such as in Millau Viaduct, Lillebælt Bridge, River Neath Bridge and also Savonius turbine in Morgenthal (2005) and Morgenthal et al. (2014).

The paper is organized as follows: Section 2 briefly describes the VPM and existing implementations. The methodology of FRM is described in Section 3. The validation of the method and computationally effective techniques are outlined in Section 4. The quality quantitative assessments are performed in Section 5 and constitute the validation of the concept. The application of flow reproduction to wake buffeting is presented in Section 6. Finally, conclusive notes are presented in Section 7.

2. The Vortex Particle Method

The Vortex Particle Method is based on the simplified vorticity description of the fundamental Navier–Stokes (NS) equation. For incompressible unsteady flow of a viscous fluid, the NS equation may be described in terms of velocity $\boldsymbol{u}(\boldsymbol{x}, t)$ as:

$$\frac{\partial \boldsymbol{u}}{\partial t}+(\boldsymbol{u}\cdot\nabla)\boldsymbol{u}=-\frac{1}{\rho}\nabla p+\nu\nabla^2\boldsymbol{u}, \tag{1}$$

where ρ is the fluid density, p is the pressure and ν is the kinematic viscosity. The continuity equation $\nabla\cdot\boldsymbol{u}=0$ must be satisfied by Eq. (1). It is found advantageous to describe the dynamics of the fluid flow in terms of the evolution of the vorticity field (Cottet and Koumoutsakos, 2000). The vorticity $\boldsymbol{\omega}$ is the curl of velocity field $\boldsymbol{u}$ of a flow, such that

$$\boldsymbol{\omega}=\nabla\times\boldsymbol{u}. \tag{2}$$

For two-dimensional flows, the NS Eq. (1) in terms of the vorticity can be written as:

$$\frac{\partial \boldsymbol{\omega}}{\partial t}+(\boldsymbol{u}\cdot\nabla)\boldsymbol{\omega}=\nu\nabla^2\boldsymbol{\omega}. \tag{3}$$

This eliminates the pressure term, which provides a lot of computational advantages. For inviscid flow Eq. (3) can be rewritten in substantial derivative notation:

$$\frac{\mathrm{D}\omega}{\mathrm{D}t}=0. \tag{4}$$

Here, Eq. (4) allows the use of a gridless numerical scheme and the discretization of particle elements in a Lagrangian manner. After applying the boundary conditions, the velocity field needs to be determined in order to evolve the flow. The velocity field can be expressed in terms of a stream function $\boldsymbol{\Psi}$ and a velocity potential ϕ such that

$$\boldsymbol{u}=\nabla\times\boldsymbol{\Psi}+\nabla\phi, \tag{5}$$

where the gradient of the potential is the free stream velocity $\nabla\phi=\boldsymbol{U}_\infty$. Taking the curl of Eq. (5) yields to Poisson equation,

$$\nabla^2\boldsymbol{\Psi}=-\boldsymbol{\omega}. \tag{6}$$

It is possible to solve Eq. (6) to compute the streamline function $\boldsymbol{\Psi}$ using the Green's function. By taking the curl of the solution $\boldsymbol{\Psi}(\boldsymbol{x})$, the velocity field in $\mathcal{R}^2$ can be computed using Eq. (7) which generally refers to the Biot–Savart relation,

$$\boldsymbol{u}(\boldsymbol{x})=\boldsymbol{U}_\infty-\frac{1}{2\pi}\int_{\mathcal{D}}\frac{\boldsymbol{\omega}(\boldsymbol{x}_0)\times(\boldsymbol{x}_0-\boldsymbol{x})}{|\boldsymbol{x}_0-\boldsymbol{x}|^2}\,\mathrm{d}\mathcal{D}_0. \tag{7}$$

The vorticity field for a domain discretized with N_p Lagrangian vortex particles can be represented as:

$$\omega(\boldsymbol{x},t)=\sum_{p=1}^{N_p}\delta(\boldsymbol{x}-\boldsymbol{x}_p(t))\Gamma_p, \tag{8}$$

where $\omega(\boldsymbol{x}, t)$ is the vorticity at position $\boldsymbol{x}$ and time t, δ is the Dirac delta function, Γ_p is the strength of pth particle, and $\boldsymbol{x}_p(t)$ is the position of pth particle in the vorticity field. The velocity of a point