Available online at www.sciencedirect.com

ScienceDirect

Procedia IUTAM 18 (2015) 1 – 7

Procedia IUTAM

www.elsevier.com/locate/procedia

IUTAM Symposium on Particle Methods in Fluid Mechanics

Vortex Particle-Mesh with Immersed Lifting Lines for Aerospace and Wind Engineering

S. Backaert[a]*, P. Chatelain[a], G. Winckelmans[a]

[a]*Institute of Mechanics, Materials and Civil Engineering (iMMC)*
Université catholique de Louvain (UCL), B-1348 Louvain-la-Neuve, Belgium.

Abstract

We present the treatment of lifting lines with a Vortex Particle-Mesh (VPM) methodology. The VPM method relies on the Lagrangian discretization of the Navier-Stokes equations in vorticity-velocity formulation. The use of this hybrid discretization offers several advantages. The particles are used solely for the advection, thereby waiving classical time stability constraints. They also exploit the compactness of vorticity support, leading to high computational gains for external flow simulations. The mesh, on the other hand, handles all the other computationally intensive tasks, such as the evaluation of the differential operators and the use of fast Fourier-based Poisson solvers, which allow the combination of unbounded directions and inlet/outlet boundaries. Both discretizations communicate through high order interpolation. The mesh and the interpolation also allow for additional advances; they are used to handle Lagrangian distortion by reinitializing the particle positions onto a regular grid. This crucial step, referred to as remeshing, guarantees the accuracy of the method. In addition, the resulting methodology provides computational efficiency and scalability to massively parallel architectures.

Sources of vorticity are accounted for through a lifting line approach. This line handles the attached and shed vorticity contributions in a Lagrangian manner. Its immersed treatment efficiently captures the development of vorticity from thin sheets into a three-dimensional field. We apply this approach to the simulation of wake flows encountered in aeronautical and wind energy applications. An important aspect in these fields is the handling of turbulent inflows. We have developed a technique for the introduction of pre-computed or synthetic turbulent flow fields in vorticity form. Our treatment is based on particles as well and consistent with the Lagrangian character of the method. We apply here our method to the investigation of wind turbine wakes over very large distances, reaching cluster or wind farm sizes.

Selection and/or peer-review under responsibility of the Technical University of Denmark, Department of Mechanical Engineering.

Keywords: Vortex method; particle mesh method; VPM; immersed lifting lines

---

1. Introduction

The envisioned developments in wind energy over the coming decade involve larger sizes for the devices and power plants, and higher efficiencies, too. These trends will make power conversion processes and the device structures

---

[*] Corresponding author. Tel.: +32-10-47-22-15 ; fax: +32-10-45-26-92
E-mail address: stephane.backaert@uclouvain.be

Selection and/or peer-review under responsibility of the Technical University of Denmark, Department of Mechanical Engineering.
doi:10.1016/j.piutam.2015.11.001

even more sensitive to flow unsteadiness and the subsequent fatigue processes. In those respects, understanding the interactions between the atmospheric turbulence and wind turbine aerodynamics will be crucial.

Wind turbine aerodynamics have been the focus of numerical investigations relying on several techniques. Free wake Vortex Lattice methods offer an affordable means of evaluating the dynamics of the vortex shedding and the near wake; they cannot however handle the dissipation of this sheet and its transition into a three-dimensional fully turbulent flow. Actuator line techniques have been proposed and include a body force term that accounts for the blade loading into a Navier-Stokes solver [1]. These techniques have been going through intensive development over recent years and have been applied to cases with a turbulent inflow [2].

Vortex methods can be seen to combine the aspects of free wake Vortex Lattice methods and actuator line techniques. Based on a vorticity formulation, they exploit the compactness of that quantity and allow the shedding of fine vortical structures [3, 4]. At the same time, they allow these flow structures to evolve into a fully turbulent flow, which makes them akin to a high fidelity variant of a free wake method.

We present work on a state-of-the-art variant of vortex methods that combines both particles and a mesh, exploiting the advantages of both discretizations. We here introduce the handling of a turbulent inflow, in vorticity form. We apply our approach to a single wind turbine, and subject it to an incoming turbulent flow which has been modelled in a preliminary Large Eddy Simulation (LES).

2. Methodology

2.1. Vortex Particle-Mesh method

We consider a three dimensional incompressible flow and the Navier-Stokes equations in their velocity ($\vec{u}$)-vorticity ($\vec{\omega} = \nabla \times \vec{u}$) form

$$\frac{D\vec{\omega}}{Dt} = (\vec{\omega}\cdot\nabla)\,\vec{u} + \nu\nabla^2\vec{\omega} \tag{1}$$

$$\nabla\cdot\vec{u} = 0 \tag{2}$$

where $\frac{D}{Dt} = \frac{\partial}{\partial t} + \vec{u}\cdot\nabla$ denotes the Lagrangian derivative and ν is the kinematic viscosity.

Vortex methods discretize the vorticity field with particles, characterized by a position $\vec{x}_p$, a volume V_p and a strength $\vec{\alpha}_p = \int_{V_p} \vec{\omega}\, d\vec{x}$. Particles are convected by the flow field and their strength is modified to account for vortex stretching and diffusion. The evolution equations of these particles read

$$\frac{d\vec{x}_p}{dt} = \vec{u}\,(\vec{x}_p) \tag{3}$$

$$\frac{d\vec{\alpha}_p}{dt} = \left((\vec{\omega}\cdot\nabla)\,\vec{u}\,(\vec{x}_p) + \nu\nabla^2\vec{\omega}\,(\vec{x}_p)\right)\, V_p. \tag{4}$$

Using the definition of vorticity and the incompressibility constraint, the velocity field, which is needed for the above particles evolution equations, is computed by solving the Poisson equation. It is obtained through the Helmoltz decomposition and reads

$$\nabla^2\vec{u} = -\nabla\times\vec{\omega}\,. \tag{5}$$

This equation is solved with the following boundary conditions: an inlet-outlet direction (along z) and unbounded in the transverse directions (along x and y).

The vortex particles-mesh method uses, in addition to the set of particles, a mesh. Combined with an high order interpolation scheme, vorticity carried by particles is interpolated onto this mesh. Differential operators (such as those for stretching and diffusion) are evaluated on the mesh using fourth order finite differences and the Poisson equation for velocity (5) is handled on the grid in Fourier space. The results of these calculations, i.e. the right-hand side of (4) and the velocity field, are then interpolated back from the grid onto the particles.

We use remeshing [5, 6] in order to remedy the loss of accuracy due to Lagrangian distortion. Remeshing consists in the periodic regularization onto a grid of the particle set via high order interpolation. In the present work, remeshing is performed at the end of each time step.

Finally, the grid-based Poisson solver is also used to reproject vorticity onto a solenoidal field, i.e. enforcing

$$\nabla \cdot \vec{\omega} = 0 \,, \tag{6}$$

every few time steps (of the order of 50 for the present simulations).

We refer to [3, 7] for details on the parallel implementation and the Poisson solver.

In order to carry out LES, the method is combined here with a hyper-viscosity subgrid-scale model [8]; the term $\nu_h(-1)^{h+1}\nabla^{2h}\vec{\omega}$ is added to the right-hand side of Eq. (1). Our implementation uses a low order $h = 2$ hyper-viscosity based on Finite-Differences. The method does not detract from other more advanced sub-grid models, such as Smagorinsky or Regularized Variational Multiscale models [9]; their implementation is a topic of ongoing work. The result is an approach that allows the accurate capture of wake dynamics with minimal spurious dispersion and diffusion and that also waives classical time stability conditions (CFL) for advection.

2.2. Immerserd lifting lines

The sources of vorticity are modeled through a lifting line model, in a fashion similar to a Vortex Lattice technique but here immersed in the mesh.

If the lift distribution along this line is known, the bound circulation $\vec{\Gamma}(r,t)$ and the shed vortex sheet can be obtained from the Kutta-Joukowsky equation and the solenoidal character of vorticity.

The method has two elements. First, the lifting line is discretized with bound vortex particles. The vorticity of these particles relies on the circulation distribution, which is determined using the Kutta-Joukowsky equation

$$\vec{L} = \rho\, \vec{v} \times \vec{\Gamma} \tag{7}$$

where $\vec{L}$ is the lift, ρ the air density and $\vec{v}$, the relative velocity. The aerodynamic performance, i.e. the lift and the drag, is computed from the airfoil polar data combined with the relative velocity, which is interpolated from the velocity field on the mesh onto these bound particles.

Then, the spatial and temporal variations of this circulation lead to the generation of a vortical sheet in the wake. In our particle-mesh setting, this shed vortical structure is actually an incremental change to an three-dimensional vorticity field. To determine the shape of this structure shed during one time step, Lagrangian tracers are used. From this shape, the vortical sheet is geometrically rebuilt, *à la* vortex lattice method (Fig. 1). We note that this procedure respects the solenoidal property of the vorticity field.

Finally, this sheet is discretized with particles, which are shed into the three-dimensional flow.

These bound and shed components can then be interpolated onto the mesh independently: both at the same time, when we need to solve (5); the shed vorticity alone, when we increment the free vorticity at the end of a time step [10].

2.3. Turbulent inflow

The implementation of a turbulent inflow has to preserve two properties of our VPM method: its Lagrangian treatment and ability to handle unbounded directions (Fig. 2).

First, we follow the approach of [11] and translate a turbulent velocity field into a set of vorticity particles, more adequate for our Lagrangian method. A buffer region feeds particles into the domain at the target mean velocity; the particles strengths are interpolated ”on the fly” from the turbulent field. The velocity of the particles lying in this region progresses from the mean velocity for the newest ones to the velocity induced by the Biot-Savart law for those ready to be added to the computational domain.

Secondly, some precautions in the unbounded directions have to be taken while this field is being fed. Indeed, the evolution of the incoming turbulence close to unbounded boundaries might cause the initial domain to grow. This would cancel the advantage of the compactness of the vorticity support. A clipping function is applied to the strengths of the entering particles in the unbounded directions. The combination of this smooth function with

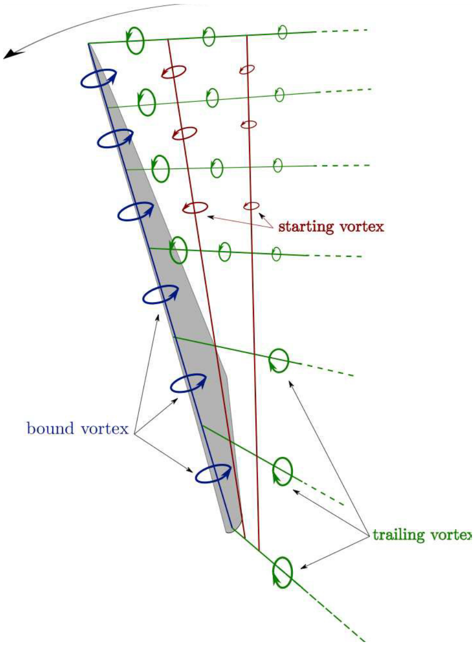


Fig. 1. Immersed lifting line: Vorticity sources modeled through a lifting line approach, here for a rotating blade

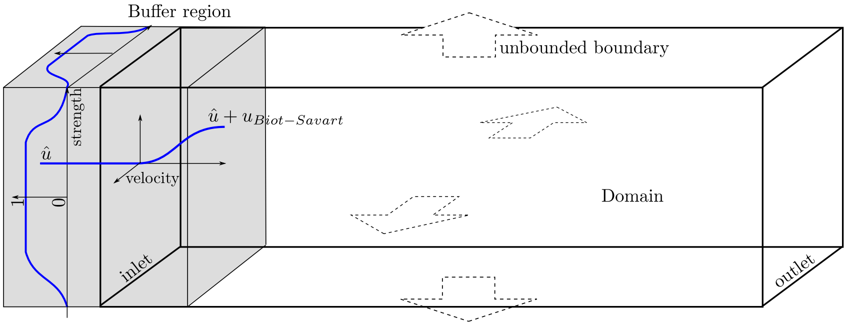


Fig. 2. Turbulent inflow and computational domain: vorticity clipping and velocity field transition in the buffer region .

the periodic reprojection of the overall vorticity field onto a solenoidal field preserves the compact support of the incoming turbulence and properties such as its isotropy or homogeneity.

The turbulent field is introduced periodically into the domain. Figure 3 shows the vorticity field generated by the feeding of a 2D long cubic homogeneous isotropic turbulent (HIT) field into a 2Dx2Dx8D domain. This HIT was generated by a spectral LES of forced turbulence [12]. We observe that the entrainment of null vorticity patches towards the center line. This calls for an initial domain large enough to push this effect away from the physics of interest.

The evolution of the characteristics of the HIT field is presented in Fig. 4. As expected, the turbulence exhibits a decay as it travels downstream. The velocity variances remain close, indicating the minor effect of the clipping on isotropy.

Other turbulent fields such as synthetic turbulent flows can be fed as well [13]. They provide more realistic unsteady wind conditions. In particular, large scales structures contained in these inflows will interact with the vortical structures shed by the lifting lines. This will lead to more pronounced vortical instabilities in the wake and low frequency oscillations of the far wake. The advection of these structures with no dispersion error is an important aspect for the wakes interaction study.

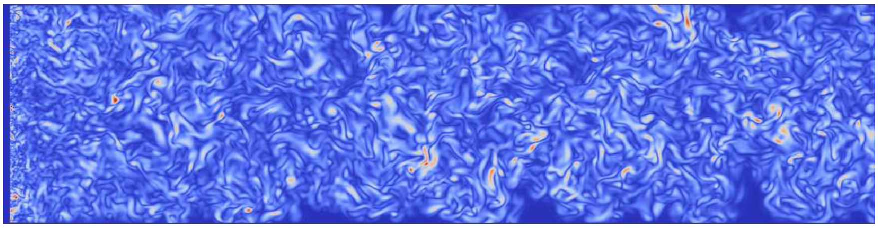


Fig. 3. Turbulent inflow: 2D cubic turbulent box fed into 2Dx2Dx8D domain, vorticity magnitude in the center plane.

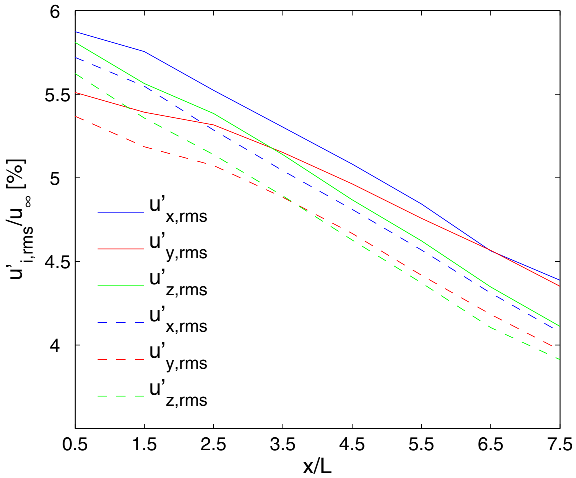


Fig. 4. Variances of velocity components. Two different sub-grid scale (SGS) models were used: a second order HV model and the Smagorinsky model. The slight difference near the inlet is due to the mismatch between the SGS models used to compute the prescribed turbulent field and the one acting during the simulation.

3. Results

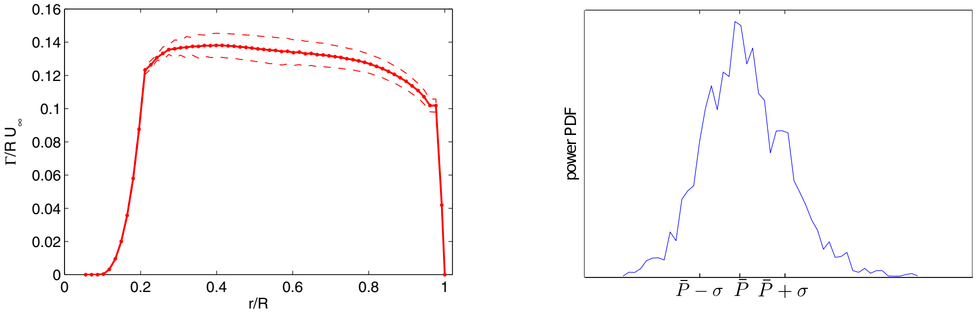
We assess our methodology on the Tjaereborg wind turbine as this has been the benchmark in several works [14, 15] with a tip-speed ratio $\lambda = \Omega R/U_\infty = 7.07$.

The computational domain covers eight rotor diameters, with the wind turbine placed one diameter from the inlet boundary. The spatial resolution amounts to 64 points per blade, leading to a initial problem size of $256 \times 256 \times 1024$ ($\sim 67 \times 10^6$ particles). This particular simulation covers 10 dimensionless times D/U_∞ and used 1024 cores for 24 hours.

The time-averaged circulation along the blades conserves the same distribution as in the case with a uniform inflow (Fig. 5(a)). However, its variance is not uniform. The contributions of the variations of relative velocity and angle of attack vary along the blade: strong variation of the angle of attack near the root and higher influence of the rotation speed near the tip.

The integration of the forces acting on each blade element leads to the power probability density function (PDF) and to the PDFs of rotor fatigue contributions such as the azimuthal bending moment or the torque variation. The figure Fig. 5(b) shows indeed the effect of this turbulence on the produced power.

The turbulent kinetic energy (TKE) profiles (Fig. 7) show that the slipstream is more turbulent in the near wake than in the case without turbulent inflow [10]. This is due to the more pronounced lateral oscillations of the tip vortices (Fig. 7). The instabilities of these structures have an impact on the length of the hub jet, which breaks down sooner.



(a) Dimensionless circulation distribution: average (solid) and standard deviation (dashed)

(b) Probability density function of power: the standard deviation σ is shown.

Fig. 5. Tjaereborg rotor: turbulence effects.

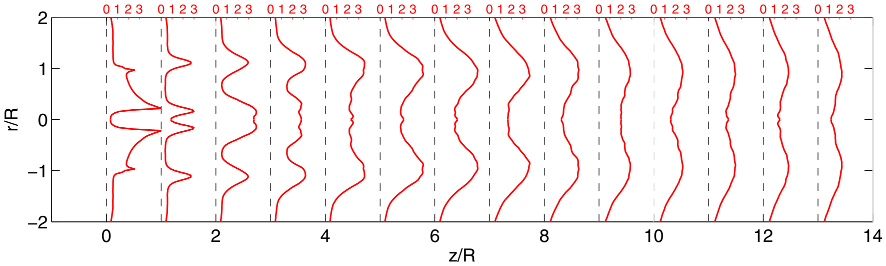


Fig. 6. Tjaereborg rotor: turbulent kinetic energy profiles $\left\langle u_x'^2 + u_y'^2 + u_z'^2 \right\rangle / 2U_\infty^2$ (%).

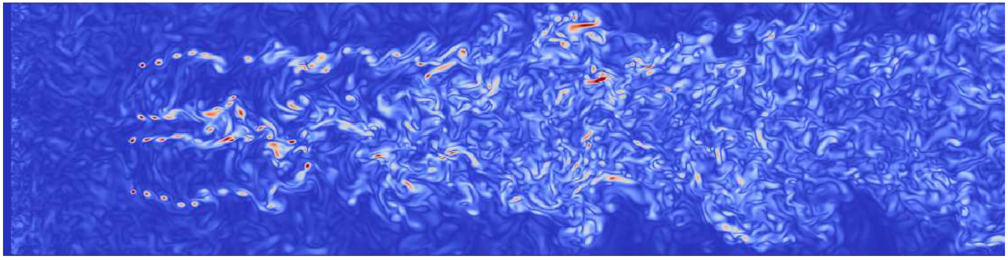


Fig. 7. Tjaereborg rotor: vorticity magnitude in center plane.

4. Conclusions

We have developed a vortex particle-mesh method combined with an immersed lifting line technique.

The discretisation of the sources of vorticity with particles allows us to account for the development of these sources into a fully developed three-dimensional flow.

The additional turbulent inflow technique, still based on a particle-based paradigm, permits simulations of wakes in unsteady wind conditions and their interactions with the sources of vorticity.

Our approach is Lagrangian and does not impose further constraints on the time step. Our results show the capability of the method to capture correctly the evolution of turbulent wakes as well as the turbulent effects on the rotor performance.

Areas of future work include multiple wind turbine investigations, specifically wake interactions and meandering. Interactions with an atmospheric boundary layer is also envisaged, although this will require the treatment of boundaries in the Poisson solver. Finally, we want to exploit this method on other wake flows such as those resulting from a vertical axis wind turbine (VAWT) or on wake-related problematics such as those encountered in formation flying.

Acknowledgements

Development work and production simulations utilized the facilities and resources of the centre de Calcul Intensif et Stockage de Masse (CISM) at the Université catholique de Louvain (UCL).

References

[1] Sorensen JN, Shen WZ. Numerical modeling of wind turbine wakes. Journal of Fluids Engineering - Transactions of the ASME. 2002 Jun;124(2):393–399.

[2] Troldborg N, Sorensen JN, Mikkelsen R. Actuator line simulation of wake of wind turbine operating in turbulent inflow [Journal Paper]. Journal of Physics: Conference Series. 2007;p. 012063 (15 pp.).

[3] Chatelain P, Curioni A, Bergdorf M, Rossinelli D, Andreoni W, Koumoutsakos P. Billion vortex particle Direct Numerical Simulations of aircraft wakes. Computer Methods in Applied Mechanics and Engineering. 2008 Feb;197(13):1296–1304.

[4] Cocle R, Winckelmans G, Daeninck G. Combining the vortex-in-cell and parallel fast multipole methods for efficient domain decomposition simulations. Journal of Computational Physics. 2008 Nov;227(21):9091–9120.

[5] Cottet GH. Artificial viscosity models for vortex and particle methods. J Comput Phys. 1996;127(2):299–308.

[6] Winckelmans G. Vortex Methods. In: Stein E, De Borst R, Hughes TJR, editors. Encyclopedia of Computational Mechanics. vol. 3. John Wiley and Sons; 2004. .

[7] Chatelain P, Koumoutsakos P. A Fourier-based elliptic solver for vortical flows with periodic and unbounded directions. Journal of Computational Physics. 2010 4;229(7):2425–2431. Available from:

[8] Borue V, Orszag SA. Local energy flux and subgrid-scale statistics in three-dimensional turbulence. Journal of Fluid Mechanics. 1998 Jul;366:1–31.

[9] Jeanmart H, Winckelmans G. Investigation of eddy-viscosity models modified using discrete filters: A simplified ”regularized variational multiscale model” and an ”enhanced field model”. Physics of Fluids. 2007 May;19(5):055110.

[10] Backaert S, Chatelain P, Bricteux L, Winckelmans G, Koumoutsakos P. Vortex particle-mesh methods with immersed lifting lines applied to the Large Eddy Simulation of wind turbine wakes. Journal of Computational Physics. in preparation;.

[11] Rasmussen JT, Hejlesen MM, Larsen A, Walther JH. Discrete vortex method simulations of the aerodynamic admittance in bridge aerodynamics. J Wind Eng Ind Aerodyn. 2010;98:754–766.

[12] De Visscher I, Bricteux L, Winckelmans G. Aircraft vortices in stably stratified and weakly turbulent atmospheres: simulation and modeling. AIAA Journal. 2012;(accepted).

[13] Mann J. The spatial structure of neutral atmospheric surface-layer turbulence. Journal of Fluid Mechanics. 1994;273:141–168.

[14] Ivanell S, Sorensen JN, Mikkelsen R, Henningson D. Analysis of Numerically Generated Wake Structures. Wind Energy. 2009 Jan;12(1):63–80.

[15] Troldborg N, Sorensen JN, Mikkelsen R. Numerical simulations of wake characteristics of a wind turbine in uniform inflow. Wind Energy. 2010 Jan;13(1):86–99.