Hybrid Eulerian-Lagrangian flow solver for multi-body fluid-structure interactions

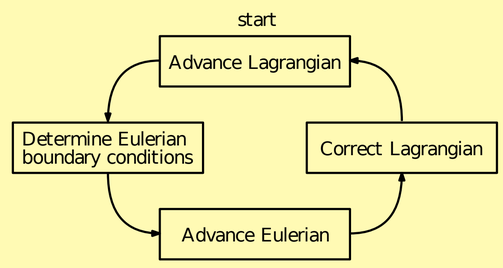
Carlos Baptista, Carlos Ferreira, Alexander van Zuijlen, Artur Palha, Gerard van Bussel

Delft University of Technology, Faculty of Aerospace Engineering, Wind Energy Research Group, Kluyverweg 1, 2629 HS Delft, The Netherlands

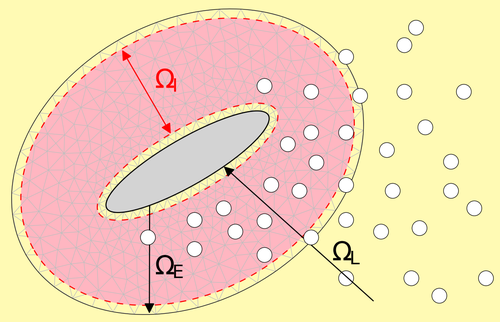
Introduction

Simulating flows around wind turbines is very costly due to multi-disciplinary complexity. An efficient approach is to decompose the flow domain into sub-domains (on basis of local flow features) and to apply the most effective method for each sub-domain. pHyFlow [1] is a hybrid Eulerian-Lagrangian framework combining a mesh-based Navier-Stokes solver (for near-body flows) and a mesh-free Vortex-Particle Method [2] (for wakes).

Hybrid coupling procedure



Coupling procedure flow diagram



Hybrid Eulerian-Lagrangian domain [1]

Advance Lagrangian

Evolve the vortex particles by one time step, while neglecting vortex generation at solid boundaries.

Determine Eulerian boundary conditions

Apply the Biot-Savart law on the vortex particles to determine the Dirichlet boundary conditions for the velocity field in the Eulerian domain.

Advance Eulerian

Evolve the Eulerian solution by one time step using any mesh-based solver.

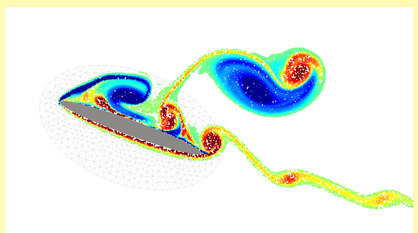
Correct Lagrangian

Use the Eulerian solution to correct the Lagrangian solution in the near-wall region.

Application: Stalled airfoil and multi-body interaction

Stalled elliptic airfoil

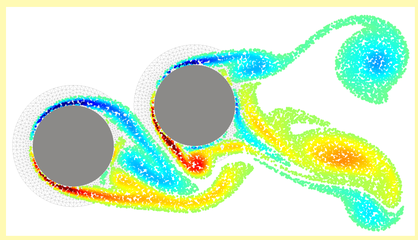
The purpose of studying the flow around an airfoil in stall is to demonstrate the capability of pHyFlow to simulate complex flow problems, similar to the flow condition for Vertical-Axis Wind Turbines. The flow around a thin elliptic airfoil with a maximum thickness of 12% chord is simulated at $\alpha = 20^o$ and $Re = 5000$. Results indicate that pHyFlow is capable of simulating unsteady flow separation. A turbulence model is, however, required to run simulations at higher Reynolds numbers in a computationally feasible manner.



Flow past stalled elliptic airfoil [1]

Multi-body interaction

pHyFlow allows the use of multiple Eulerian domains. This facilitates mesh generation for multi-body simulations as each body can be meshed independently. The purpose of studying the flow around multiple cylinders is to demonstrate the capability of simulating multi-body interactions and to explore the application of segragated meshes. The feasibility of simulating the flow around two cylinders using segragated meshes leads to the next step of simulating the flow around multiple moving bodies. This will be explored in the future.



Flow past two cylinders using segragated meshes [1]

References

[1] Artur Palha, Lento Manickathan, Carlos Simao Ferreira and Gerard van Bussel. A hybrid Eulerian-Lagrangian flow solver, 2015. arXiv:1505.03368v2.

[2] Georges-Henri Cottet and Petros D. Koumoutsakos. Vortex methods - Theory and Practice. Cambridge University Press, 2000.

[3] FP7 AVATAR (AdVanced Aerodynamics Tools for lArge Rotors) project. http://www.eera-avatar.eu, 2013.

Governing equations

Eulerian flow solver

The Eulerian solver follows a mesh-based approach using either a Finite-Element method (based on FEniCS) or a Finite-Volume method (based on OpenFOAM) to solve the incompressible Navier-Stokes equations in the (u, p) formulation:

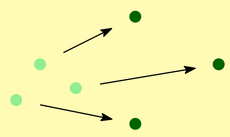
$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u}$$

Lagrangian flow solver

The Lagrangian solver follows a mesh-free approach using a regularised Vortex-Particle method to solve the incompressible Navier-Stokes equations in the (u, ω) formulation:

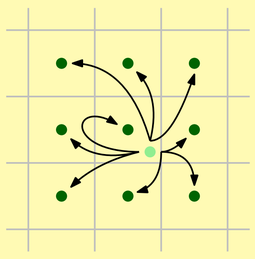
$$\frac{\partial \omega}{\partial t} + (\mathbf{u}\cdot\nabla)\omega = \nu\nabla^2\omega$$

Convection and diffusion of the particles are treated separately by applying Chorin's Viscous-Splitting method:



$$\frac{d\mathbf{x}_p}{dt} = \mathbf{u}(\mathbf{x}_p, t) \qquad \frac{d\omega}{dt} = 0$$

convection sub-step



$$\frac{d\mathbf{x}_p}{dt} = \mathbf{0} \qquad \frac{d\omega}{dt} = \nu\nabla^2\omega$$

diffusion sub-step

Current work: Interfacing with OpenFOAM

Currently, pHyFlow uses an Eulerian flow solver based on the Finite-Element library of FEniCS. RANS models are needed to facilitate the simulation of high-Reynolds turbulent flows. However, the lack of validated RANS models in FEniCS promts the migration to a different solver. Therefore, current work revolves around replacing FEniCS by another Eulerian solver based on the Finite-Volume library of OpenFOAM. Interfacing pHyFlow with OpenFOAM enables the framework to exploit, among others, the well-validated Spalart-Allmaras and k - ω SST RANS turbulence models.

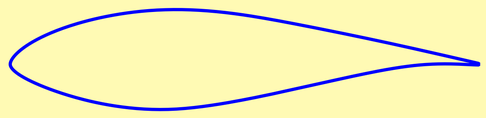
Outlook: Towards high-Reynolds flows in 3D

Verification

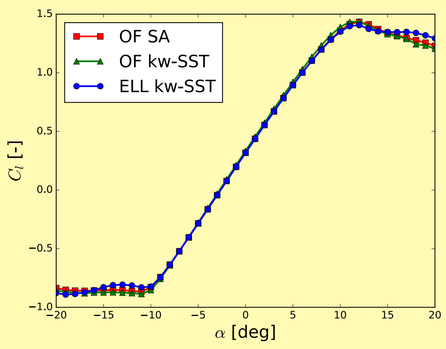
After the interfacing of pHyFlow with OpenFOAM has been completed, the new configuration of the hybrid framework will be tested against the old configuration which uses FEniCS. The flows past an impusively started cylinder and a stalled elliptical airfoil will be used for the verification of the new configuration as these cases were previously used to validate the old configuration.

Validation

As part of the FP7 AVATAR project [3], full-Eulerian simulations of a steady DU00-W-212 airfoil, using OpenFOAM, have been performed for Reynolds numbers up to 15 million. The resulting 2D lift and drag polars had a good match with results from parters in the AVATAR poject with their in-house codes. These results will therefore be used as reference data for an initial validation of the hybrid framework for high-Reynolds turbulent flows around a profile relevant for the Wind Turbine industry.



DU00-W-212 airfoil



Extension to 3D

Although, OpenFOAM is capable of both 2D and 3D simulations, the current implementation of the Vortex-Particle method (VPM) in pHyFlow is in 2D. To progress towards 3D simulations, either the implementation has to be extended to 3D or an existing freely-available 3D VPM solver will be used to replace the current 2D implementation.

TU Delft