3D Unsteady CFD model for Multi-Rotor Multi-Body simulations with OpenFOAM including Body-Vortex Interaction

Delft University of Technology, Faculty of Aerospace Engineering, Wind Energy Research Group, Kluyverweg 1, 2629 HS Delft, The Netherlands
Rention Pasolari M.Sc.,
Promotor & Daily Supervisor: **Prof.dr.ir. CJ (Carlos) Simao Ferreira**, Co-Promotor & Daily Supervisor: **Dr. Alexander van Zuijlen**

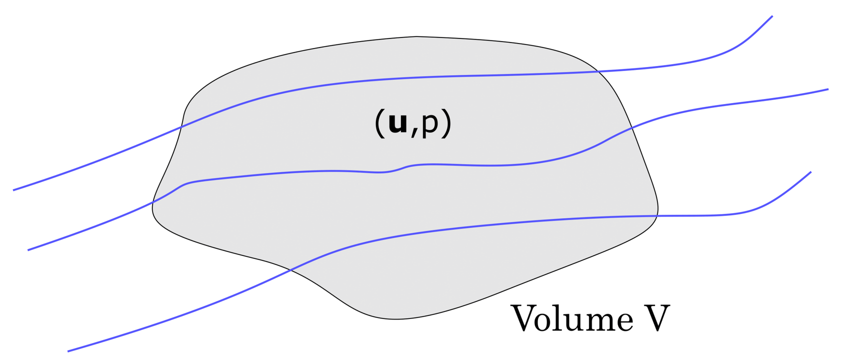
Duration of the PhD: **2020-10-01 to 2024-09-30**

Introduction

As the wind energy technology, and renewable sources of energy in general, are becoming essential in our lives, the research on these fields is extremely important. Hence, we need efficient, reliable and accurate tools to study fluid dynamic topics. It is generally known that the most popular methods used in CFD are based on *Eulerian* and *Lagrangian* approaches. Each of them has its own advantages and drawbacks. Our aim is to couple an Eulerian solver (*OpenFOAM*) with a Lagrangian solver (*Vortex Particle Method*) into *pHyFlow*, an accurate, precise and efficient CFD tool.

Eulerian Solver

The Eulerian approach focuses on specific locations in the space through which the fluid flows as time passes.



Eulerian solvers are mesh-based solvers, which use finite objects such as *Finite Differences*, *Finite Volumes* and *Finite Elements*, to solve the equations that govern the physical problem.

Advantages

- Very efficient and accurate in resolving near-solid domains

Disadvantages

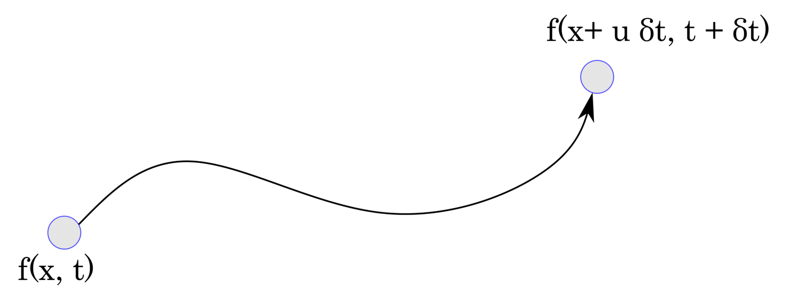
- Not accurate in resolving far-field wakes
- Mesh based solvers
- Diffusive

In *OpenFOAM* we solve the Navier-Stokes equations in the *Velocity-Pressure* form.

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

Lagrangian Solver

Lagrangian specification of the flow field is a way of looking at fluid motion where the observer follows an individual fluid parcel as it moves through space and time.



In this project, the *Vortex Particle Method* will be the Lagrangian solver, where vortex particles which carry circulation, describe the flow's properties and evolution.

Advantages

- Very efficient in describing the evolution of the vorticity in the far-field
- They allow the superposition of elements
- FMM and GPU's for acceleration of the simulation
- Mesh-free

Disadvantages

- Not accurate in resolving near-solid phenomena
- Need additional methods (Vortex Panel Methods) for the boundary conditions

The Navier-Stokes is formulated in the *Velocity-Vorticity* form, and splitted into the convection and the diffusion sub-steps.

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

Convection step

$$\frac{d\boldsymbol{x_p}}{dt} = \boldsymbol{u}(\boldsymbol{x_p})$$
$$\frac{d\alpha_p}{dt} = 0$$

Diffusion step

$$\frac{d\boldsymbol{x_p}}{dt} = 0$$
$$\frac{d\alpha_p}{dt} = \nu \Delta \alpha_p$$

Hybrid Solver

The Hybrid solver (*pHyFlow*) will exploit the advantages of each solver. For this, the whole domain is decomposed into the Eulerian and the Lagrangian part. The Eulerian subdomain is near the solid boundary in order to capture the viscous phenomena and the vorticity generation. The Lagrangian method solves the entire domain but only the solution for the far-field is kept. Inside the interpolation region, the Eulerian solution is transferred to the Lagrangian.

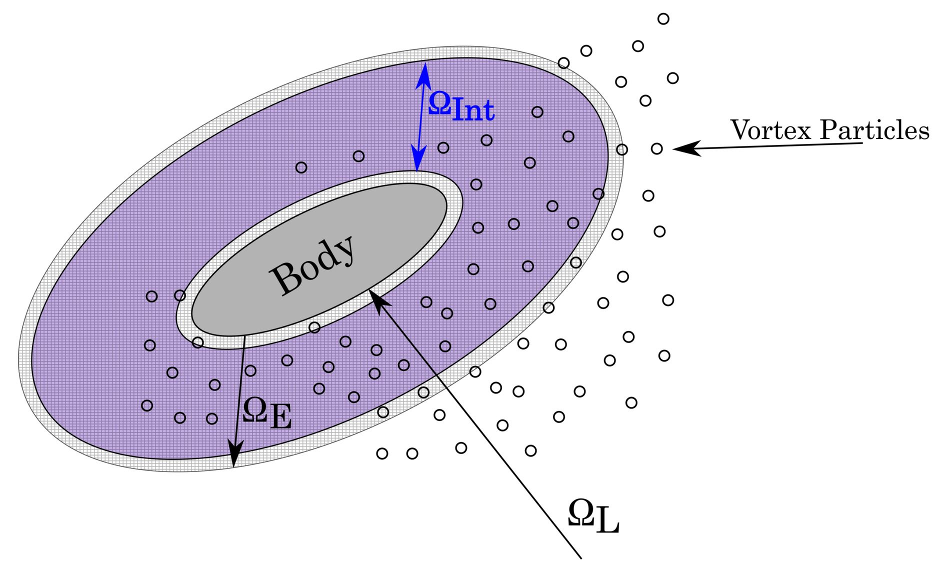
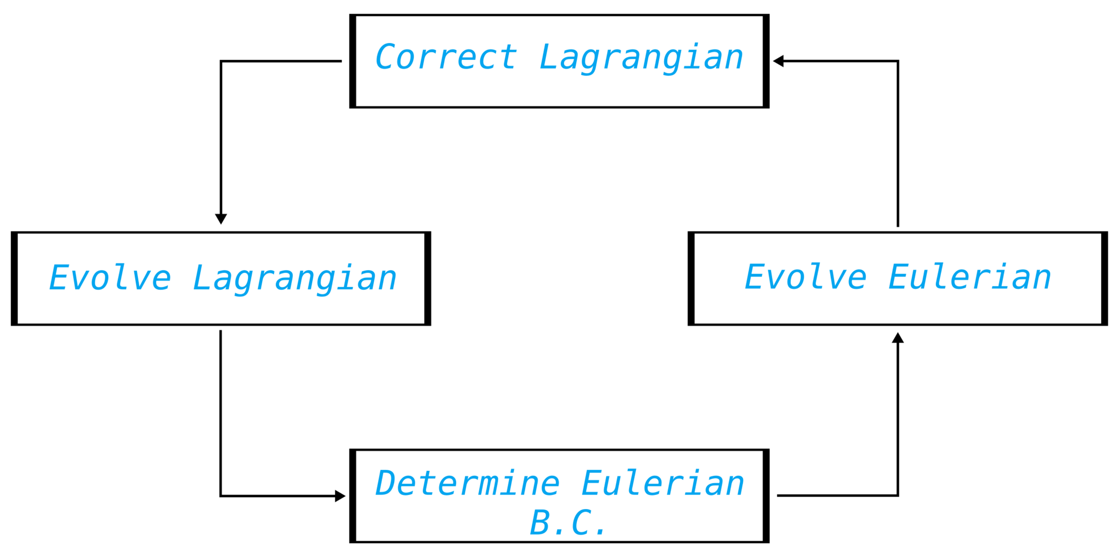


Figure: Domain decomposition in the Hybrid solver.

The steps are followed by the *pHyFlow* are:



1. Correct the Lagrangian field, using the solution of the Eulerian into the interpolation area.
2. Evolve the Lagrangian solution from t to $t + \Delta t_L$.
3. Determine the boundary conditions for the outer Eulerian boundary, using the solution of the Lagrangian.
4. Evolve the Eulerian solution from t to $t + \Delta t_L$ using k_E sub-steps.

Current work and aims

A 2D version of the *pHyFlow*, coupling the *Vortex Particle Method* with the *Finite Elements Method* using the *FEniCS* software, has already been developed by Pahla [1].

The specific Ph.D. program focuses on the development of an 3D version of the *pHyFlow*, which will use the *Finite Volumes Method* implemented in *OpenFOAM* instead of the *Finite Elements Method*.

The first step is to develop the 2D model, and validate it with the corresponding results of the available version of *pHyFlow* and other cases from bibliography, and then extend it to its 3D version. The *pHyFlow* will be able to deal with problems which are:

- Three dimensional
- Multi-Body, Multi-Rotor
- Unsteady
- Turbulent

References

- Artur Palha et al. "A hybrid Eulerian-Lagrangian flow solver". In: (2015).
- Jorge Mario Tamayo-Avendaño. "Coupling of a Finite Volume solver to a Hybrid Lagrangian-Eulerian Vortex Particle Code". In: 8 (2017).
- Lento Manickathan and Artur Palha. "Hybrid Eulerian-Lagrangian vortex particle method". In: *Msc, Technical University of Delft* (2014).
- Bjarnthor Egilsson. "Hybrid Vortex Models in 2D". In: (2015).

TU Delft — Delft University of Technology

Wind Energy Research Group