INORE European Symposium, 11-17 May 2014, La Vega, Spain

Modeling the Complex Dynamics of the Oscillating-Foils Hydrokinetic Turbine Using an Overset Mesh Technique

E.Gauthier, T.Kinsey, G. Dumas
Department of Mechanical Engineering, Laval University, Quebec City (Qc), Canada
Contact: etienne.gauthier.4@ulaval.ca

Laboratoire de Mécanique des Fluides Numérique
www.lmfn.ulaval.ca

INTERNATIONAL NETWORK ON INORE OFFSHORE RENEWABLE ENERGY

Context

The growing demand in energy coupled with the existing environmental concerns motivate the development of alternative power-generation systems using renewable energies. A novel prototype of hydrokinetic turbine based on oscillating hydrofoils (Hydrolienne à ailes oscillantes HAO) has been under development for several years at the Laboratoire de Mécanique des Fluides Numérique (LMFN) of Laval University.

To reliably predict the performances of the HAO hydrokinetic turbine, a complete representation of the system has to be numerically simulated in order to study the impact of multi-body interactions. The simulation of multiple bodies in relative motion represents a challenge for CFD simulations and meshing strategies. In this work, we rely on overset mesh techniques to overcome this challenge. The optimization of the frame shape in order to increase the hydrokinetic turbine performance is also presented to show an example of study that became realizable using overset interfaces.

Definitions

The hydrofoil motion is defined as a combination of a heaving motion and a pitching motion.

Pitching velocity
$\Omega(t) = \theta_0\,\omega\,\cos(\omega\,t)$

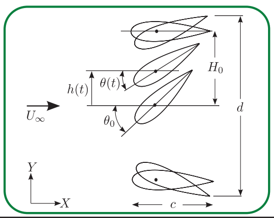
Heaving velocity
$V_y(t) = H_0\,\omega\,\cos(\omega\,t + \phi)$

Angular frequency
$\omega = 2\,\pi\,f$

$$P = \text{heaving velocity} \times \text{vertical force} + \text{pitching velocity} \times \text{moment at pitching axis}$$

Power coefficient
$$C_P = \frac{P}{\frac{1}{2}\,\rho\,U_\infty^3\,b\,c}$$

Horizontal force coefficient
$$C_X = \frac{X}{\frac{1}{2}\,\rho\,U_\infty^2\,b\,c}$$



$Re = 500\,000$

$\theta_0 = 80°$

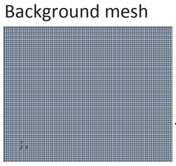
$H_0 = 1\,c$

$b =$ foil span

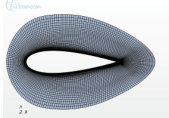
Overset mesh approach

The overset mesh approach, also known as chimera mesh technique is an efficient tool to simulate multiple bodies in relative motions. This approach has been successfully developed in CD-adapco StarCCM+ CFD solver. The overset allows the simulation of moving bodies without having to use costly techniques such as deforming mesh and remeshing.

Background mesh

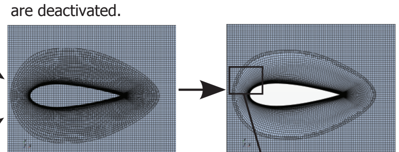


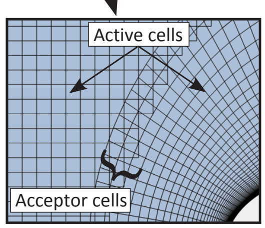
Overset mesh



Initializing the interface

The background cells behind the overset mesh region are deactivated.

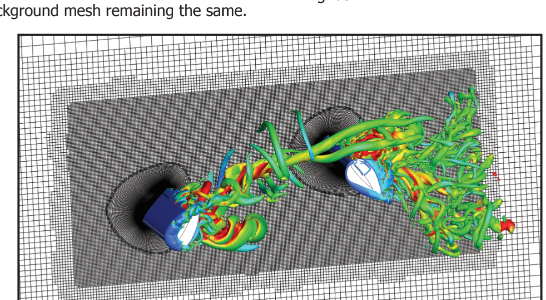




Acceptor cells are used to couple solutions on the two overlapping grids.

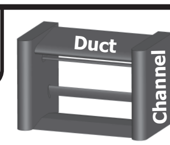
This meshing technique can be used to simulate complex motions such as those involving interactions between moving bodies in close proximity or intersecting paths and those with large motions.

This method simplifies the meshing process in the case of parametric studies involving shape optimization. Testing multiple geometries requires only to modify the superposed mesh, the background mesh remaining the same.

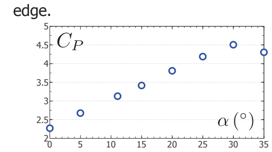


Mesh representation of hydrofoils oscillating in a tandem configuration with contours of Q-criterion colored by the turbulent viscosity ratio.

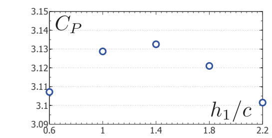
Optimizing the frame shape



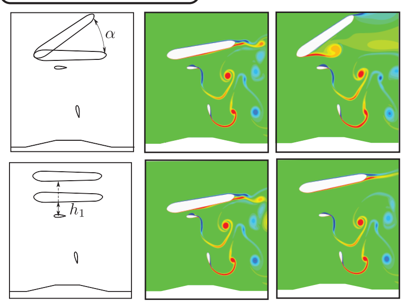
Increasing the α angle tends to increase the mass flow through the turbine and thereby the power output until stall occurs on the duct leading edge.



The distance h_1 between the duct and the maximum position of the foil has a small but noticeable impact on the power output.

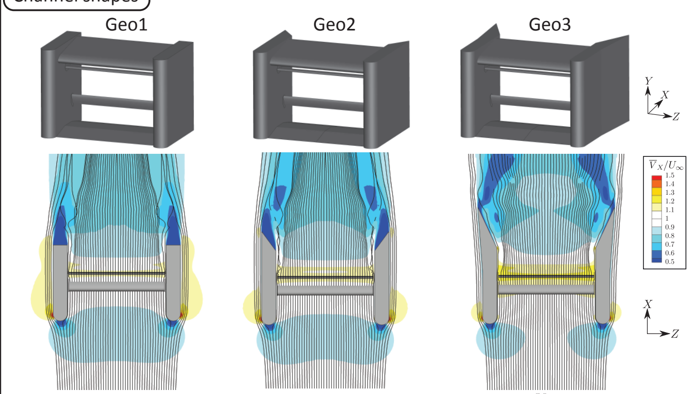


Divergent duct shapes

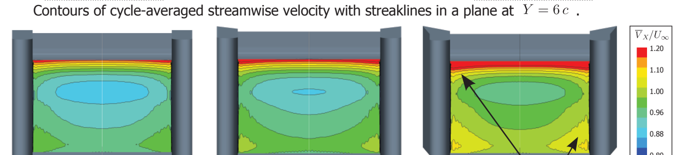


Vorticity fields (positive:red, negative:blue)

Channel shapes



Contours of cycle-averaged streamwise velocity with streaklines in a plane at $Y = 6\,c$.

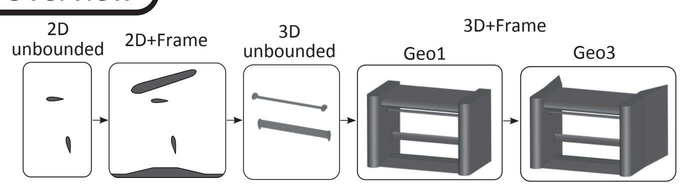


Contours of cycle-averaged streamwise velocity entering the turbine.

	C_P	$\overline{V}_{X\,section}/U_\infty$	$C_{X\,total}$
Geo1	2.48	0.96	0.259
Geo2	2.65	0.99	0.267
Geo3	3.12	1.04	0.276

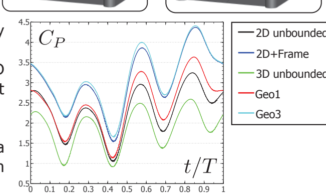
Using a divergent shape on both duct and channel increases the flow velocity through the turbine especially close to the frame walls.

Overview



Modeling the frame structure influences significantly the performances predicted. A complete representation of the turbine (including the 3D frame structure) has to be simulated in order to get good performance predictions.

Optimizing the frame shape allows us to get a power output comparable to the one obtained with a 2D simulation including the frame.



Project website: http://www.hydrolienne.fsg.ulaval.ca/

Financial support from the Natural Sciences and Engineering Research Council of Canada (NSERC), Les Fonds Québécois de la Recherche sur la Nature et les Technologie (FQRNT) and the Leadership and Sustainable Development Scholarships program of Laval University is gratefully acknowledged. Computations were made on the supercomputers Colosse from Laval University and Guillimin from McGill University, managed by CalculQuébec and Compute Canada.