FOR APPLICATION, PLEASE CONTACT ADVISOR(S) BY EMAIL WITH COPY TO:
ali.siadat@ensam.eu AND yvon.velot@ensam.eu

Research Topic for the ParisTech/CSC PhD Program

Field: Materials Science, Mechanics, Fluids

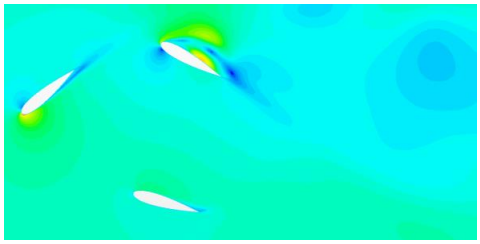
Subfield: Mechanical Engineering – Fluid Dynamics

Title: Numerical simulation, analysis, and optimization of an innovant trochoidal propeller
ParisTech School: Arts et Metiers, Campus of Lille

Advisor(s) Name: Annie-Claude Bayeul-Lainé & Olivier Coutier-Delgosha
Advisor(s) Email: annie-claude.bayeul-laine@ensam.eu & Olivier.coutier@ensam.eu

(Lab, website): LMFL (Laboratoire de Mécanique des Fluides de Lille)
http://lmfl.cnrs.fr/en/home/

Short description of possible research topics for a PhD: The operation of a trochoidal naval propeller is significantly more complex than classical propellers. Indeed, it is based on the rotation of several hydrofoils around a vertical axis, while each foil is also submitted to a periodic oscillation around its own axis. This kinematics is derived from the sculling movement, when a single oar is used to propel a ship by moving it over the stern. Its main advantage is an improved ship manoeuvrability, since it provides a thrust in any direction with a short time response. To maximize the efficiency, a detailed understanding of the internal flows is required. Modern CFD solvers, coupled with up to date optimization methods, offer promising capabilities on that level. An experimental activity is conducted at IRENav (French Naval Academy), while the numerical approach, which is the purpose of the present proposal, is performed at Art et Metiers Paristech. The present project is focused on the optimization of the foil kinematics and geometry. The objective is to obtain the maximum available hydraulic efficiency, based on the understanding of the multi-scale unsteady mechanisms in the rotor. CFD simulations will be performed with a URANS (Unsteady Reynolds Averaged Navier-Stokes) solver, as well in three dimensions (3D) in order to quantify precisely the propeller performance, as in 2D to conduct the optimization process. The influence of several parameters like the non-homogeneity of the upstream flow, the occurrence of cavitation, and the confinement (due for example to the vicinity of the ship hull) will be studied.



Required background of the student: fluid mechanics

A list of 5(max.) representative publications of the group: (Related to the research topic)

AC Bayeul-Lainé, S. Simonet, G. Bois: *Unsteady flow field in a mini VAWT with relative rotation blades: Analysis of temporal results*. IOP Conference Series Materials Science and Engineering 09/2013; 52. DOI:10.1088/1757-899X/52/5/052002

O. Coutier-Delgosha, F. Deniset, JA. Astolfi & JB. Leroux (2007), Numerical prediction of the cavitating flow on a two-dimensional symmetrical hydrofoil and comparison with experiments, *J. of Fluids Eng.* 129(3):279–292.