Strongly-coupled fluid-structure-interaction with large deformations on a flexible hydrofoil under forced oscillation

Stefan Hoerner, Cyrille Bonamy, Thierry Maître, Olivier Cleynen, Dominique Thévenin

STRONGLY-COUPLED FLUID-STRUCTURE-INTERACTION WITH LARGE DEFORMATIONS ON A FLEXIBLE HYDROFOIL UNDER FORCED OSCILLATION

STEFAN HOERNER[1], CYRILLE BONAMY[2], THIERRY MAÎTRE[3],
OLIVIER CLEYNEN[4], DOMINIQUE THÉVENIN[5]

[1] *Stefan Hoerner, LSS, University Magdeburg & LEGI, Grenoble-INP, stefan.hoerner@ovgu.de*
[2] *Cyrille Bonamy, LEGI, CNRS , cyrille.bonamy@legi.cnrs.fr*
[3] *Thierry Maître, LEGI, Grenoble-INP, thierry.maitre@legi.grenoble-inp.fr*
[4] *Olivier Cleynen, LSS, University Magdeburg, olivier.cleynen@ovgu.de*
[5] *Dominique Thévenin, LSS, University Magdeburg, thevenin@ovgu.de*

Keywords: *Fluid-Structure Interaction, Turbomachinery*

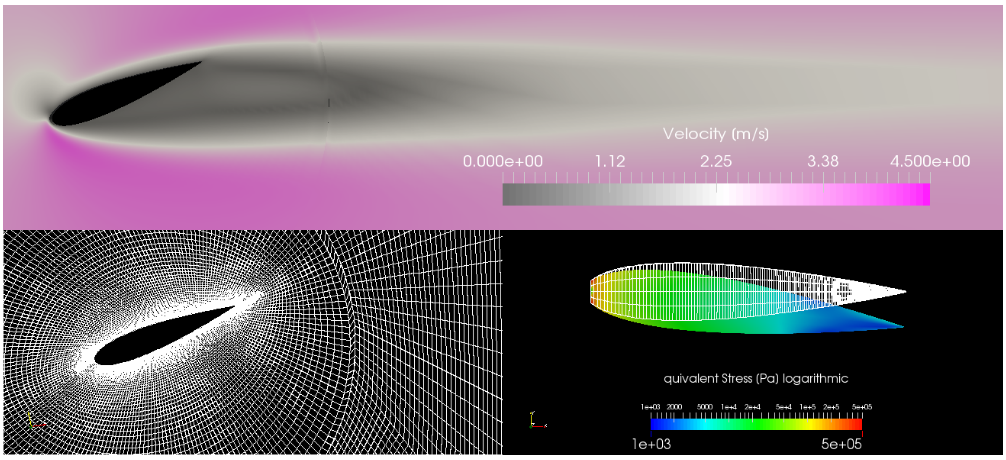
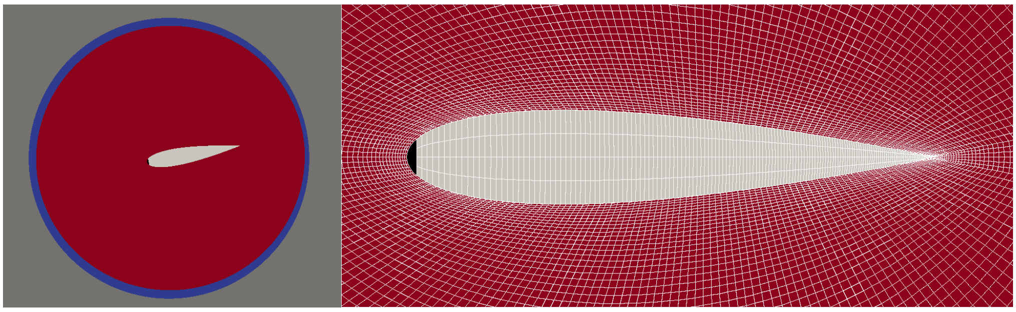


Figure 1: FSI simulations

Adaptive structures are commonplace in nature. Their flexibility prevents overload and improves motion efficiency, as has been well documented for insect wings or whale fins (*Fish* 1993,[1]). In terms of engineering, Fluid-Structure Interaction (FSI) is not only in bionics an interesting task. Rotating-machinery, like wind turbines with large, flexible composite blades, encounter large deformations. The ability to predict those and their influence on power extraction could help improve efficiency and durability of the structures. Nevertheless, such an FSI simulation is a challenging task.

In general there are two different approaches: monolithic and partitioned. The monolithic approach uses a single solver and mesh for both the fluid and the solid domain. Conversely, partitioned algorithms use at least one mesh for the fluid and one for the solid region, as well as multiple solvers. Information on

the boundary conditions has to be transfered at the interface between the two meshes. Regarding the deformation of the solid and its influence on the fluid flow, different coupling strategies are possible. The first possibility is a one-way coupling. Here, the fluid-induced pressure and shear forces are transfered as a boundary condition for the deformation in the solid region. This method works without internal FSI loops to estimate the equilibrium between solid stress and fluid-induced forces, however, it is only suitable for small deformations or stress calculations. In case of large structural deformations which influence the fluid flow significantly, a two-way coupling is necessary to achieve satisfactory results. In this case information about the solid deformation is transfered from the solid region to the fluid mesh, commonly in the form of a deformation velocity. This requires an internal FSI loop for each time step.
In 2014, *Ẑeljko Tuković et al.* contributed a Fluid-Solid Interaction toolkit to the foam-extend project [2]. It contains a strongly-coupled solver with partitioned approach with different couplings, like Aitken relaxation or the IQN-ILS method; this toolkit serves as the base for the author's contribution.



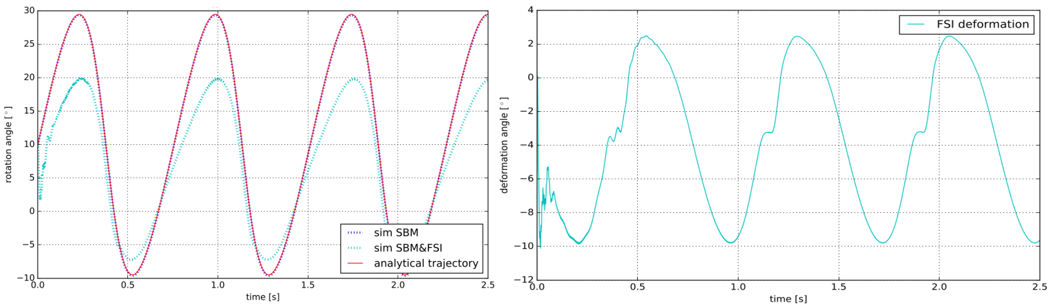
(a) Case setup with different mesh regions

(b) Detail foil

Figure 2: Case setup

The present work aims at extending the existing libraries. A functionality that allows FSI simulations of moving bodies following a forced rotational movement is implemented. This is required to study a wide range of physical problems, such as simulations in turbomachinery. Two different motion solver techniques are combined: 1) The prescribed motion of solid body, following an already-known motion law, and additionally 2) the deformation of the solid due to the fluid-induced forces. The latter are taken into account by a second motion solver, that calculates the velocity diffusion for a moving boundary in a mesh with fixed boundaries.
The fluid mesh (see figure 2a) is divided into a static (grey) and a rotating (blue/red) section, in the present case an oscillating region. A classical rotor/stator setup is implemented, using the General Grid Interface (between blue and grey) to interpolate pressure and velocity between the two meshes. In a second step, a subset of the fluid mesh is defined. In this subset, a motion solver from the *meshMotion* class is used. It is based on the standard *velocityLaplacian* motion solver, with modifications to deform only predefined parts (red) of the rotor region. In the rotor region, an additional prescribed arbitrary oscillation is performed that moves all points of the rotating region. The solid-body motion solver is inherited from the *dynamicFvMesh* class and is inspired by the motion solver developed by *Bousquet* 2012 [3] during the Open Source CFD course at Chalmers University. This solver reads the initial mesh points, obtains the deformation from a point velocity field calculated by the mesh motion solver in the subset, and adds the prescribed solid-body motion as a function of time. The highly flexible part of the solid (white) is modeled with a uniform isotropic Young's Modulus. It is attached at the non-deformable head clamped by a fixed displacement boundary condition (see figure 2b). The solid mesh itself is not deformed during the calculations, due to its total Lagrangian description; in this case the deformation is part of the solution

and provided as a vector field. Furthermore, the solid mesh does not perform a solid-body motion. Body forces, centrifugal forces are neglected in the present case. To validate the functionality of the approach,



(a) absolute FSI deformation and solid body motion (b) FSI deformation in oscillating reference frame

Figure 3: Motion at the sliding Interface and at the trailing edge

multiple simulations are performed (see figure 1). An experimental validation using the hydrodynamic tunnel at the LEGI Grenoble is planned in a later stage of the project. The simulation setup consists of a NACA0018 hydrofoil following a forced non-sinusoidal oscillation with offset in a deep stall flow regime. The deformation at the trailing edge is monitored, once in the absolute (see figure 3a) and once in the oscillating reference frame (see figure 3b). In the next step the setup will be modified with a orthotropic, multi-material modeling, to investigate the behavior of a composite hydrofoil.
The source code of the adaption including a tutorial will be published at the end of the author's thesis. Post processing was done using Python-scripts including the pyof/fluidFoam package published by *Bonamy et al*. 2017.

Acknowledgments

The authors thank the Rosa-Luxemburg-Foundation, Berlin and the German-French-University, Saarbrücken for the funding of the first authors thesis in "cotutelle franco-allemande". All the computations presented in this paper were performed using a Intel Xeon based Workstation and on the Froggy platform of the CIMENT infrastructure (https://ciment.ujf-grenoble.fr), which is supported by the Rhône-Alpes region (GRANT CPER07_13 CIRA), the OSUG@2020 labex (reference ANR10 LABX56) and the Equip@Meso project (reference ANR-10-EQPX-29-01) of the program "Investissements d'Avenir" supervised by the Agence Nationale pour la Recherche.

References

[1] F. E. Fish, "Power output and propulsive efficiency of swimming bottlenose dolphins (tursiops truncatus)," *Journal of Experimental Biology*, no. 183, pp. 179–193, August 1993. (document)

[2] Tukovic Z., Cardiff P., Karac A., Jasak H., Ivankovic A., "Openfoam library for fluid structure interaction," in *OpenFoam Workshop 9 - Zagreb*, 2014. (document)

[3] G. Bousquet. (2012) Simulation of large and complex motions using the ggi interface. CHALMERS UNIVERSITY OF TECHNOLOGY - Dept of Thermo and Fluid Dynamics. [Online]. Available: http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2012/ (document)