Development of Multi-Physics Coupled CFD solvers in OpenFOAM®

JengHei Chow, PhD, NTU

Abstract

Statement of the Problem: A huge array of CFD tools are currently available in the market today providing solutions to many problems. Commercial software is usually easy to operate but difficult to manipulate, especially when there are key physics that needs to be implemented (Keyes, et. al., 2012). For example, the last decade saw a rise in fluid-structure interaction (FSI) code development (Seng, 2012), (Chow & Ng, 2016). The requirement for a stable dynamic mesh solver pushed developers away from commercial code and towards an improved, customised solution. Customised code provides stability hacks to problems in unique ways, such as damping of the artificial added mass effect in partially submerged floating structures with dynamic under-relaxation (Chow et. al., 2019).

OpenFOAM® is a development tool for the customisation of numerical solvers to solve CFD problems. The open-source code, written in C++, allows for low level memory manipulation to optimise efficiency, and end users can reuse or build routines without having to write basic functions from scratch. OpenFOAM® is released free for public use under the GNU General Public License and over the years, the user base increased tremendously and so did the number of customised solvers, applications and utilities. The key strength of OpenFOAM® can be exhibited by its flexibility as with most "sandbox" environments, where users with pre-requisite knowledge of the underlying physics and programming techniques are free to modify existing solvers to extend their capabilities for more complex projects. The fundamental source of the solvers dates back to the original development by Jasak (1996), where a framework for modelling fluid flow applications with the Finite Volume Method (FVM) was first set up.

Figure 1 below showcases the implementation of a custom solver that can couple a certain behaviour with the CFD fluid solver.

Image

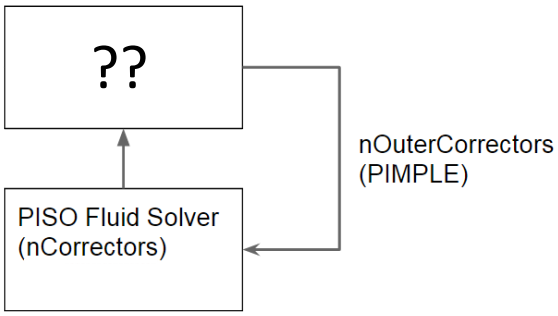


Figure 1 Schematic of a strongly coupled partitioned solver in OpenFOAM, the box signifies the physics of interest which can be motion, deformation or any variable that could be affected by the fluid dynamics over time

Recent Publications

1. CHOW, J. H. & NG, E. Y. K. 2016. Strongly coupled partitioned six degree-of-freedom rigid body motion solver with Aitken's dynamic under-relaxation. *International Journal of Naval Architecture and Ocean Engineering*, 8, 320–329.
2. CHOW, J. H., NG, E. Y. K. & SRIKANTH, N. 2019. Numerical Study of the Dynamic Response of a Wind turbine on a Tension Leg Platform with a coupled partitioned six degree-of-freedom rigid body motion solver. *Ocean Engineering*, 172, 575–582.
3. JASAK, H. 1996. Error analysis and estimation for the finite volume method with applications to fluid flows. London: Imperial College of Science, Technology and Medicine.
4. KEYES, D. E., MCINNES, L. C., WOODWARD, C., GROPP, W. D., MYRA, E. & PERNICE, M. 2012. Multiphysics Simulations: Challenges and Opportunities.
5. SENG, S. 2012. Slamming and whipping analysis of ships, Lyngby, DTU Mechanical Engineering: DCAMM.



Biography

JengHei Chow is an experienced enthusiast specialising in Computational Fluid Dynamics (CFD) applications and development of custom numerical software. He completed his industrial Ph.D. with NTU while working with DHI under the EDB Industrial Post-graduate Program (IPP). His notable work in DHI involves development of the coupled fluid-motion solvers in OpenFOAM. The developed code is currently owned by the company to measure the response of floating offshore wind turbines and other floating platforms under different wave conditions. His work contributed to two publications describing the improvements in numerical stability with customized coupled fluid-motion solvers. He is also interested in developing custom solvers for innovative projects related to water and environment such as indoor air, heat and moisture modelling.

Email: chowjenghei@gmail.com

Notes/Comments: