International Journal of Offshore and Polar Engineering (ISSN 1053-5381)

Vol. 32, No. 1, March 2022, pp. 24–30; https://doi.org/10.17736/ijope.2022.jc844

http://www.isope.org/publications

A High-performance Open-source Solution for Multiphase Fluid-Structure Interaction

Wendi Liu, Stephen M. Longshaw, Alex Skillen and David R. Emerson
Scientific Computing Department, Science and Technology Facilities Council, Daresbury Laboratory
Warrington, United Kingdom

Carmine Valente and Francesco Gambioli
Airbus Operations Ltd.
Filton, Bristol, United Kingdom

A multiphase fluid-structure interaction (FSI) framework using open-source software has been developed, utilising components able to run on high-performance computing platforms. A partitioned approach is employed, ensuring a separation of concerns (fluid, structure, and coupling), allowing design flexibility and robustness while reducing future maintenance effort. Multiphase FSI test cases have been simulated and compared with published results and show good agreement. This demonstrates the ability of this multiphase FSI framework in simulating complex and challenging cases involving a free liquid surface.

INTRODUCTION

An important phenomenon in a wide range of scientific and engineering disciplines is the interaction between multiphase flow and elastic structures, such as an aircraft wing with a sloshing fuel tank (Gambioli et al., 2019, 2020; Mastroddi et al., 2019, 2020; Titurus et al., 2019; Saltari et al., 2021) and the impact of ocean waves on elastic ocean structures (Gomes et al., 2020). Accurately simulating a multiphase fluid-structure interaction (FSI) can help reveal the mechanisms behind important and complex real-world phenomena, allowing for important design considerations, such as how to protect an elastic structure from fatigue or failure (Botha and Hindley, 2015) or how to achieve active/passive control of a system (Ducoin et al., 2012). There is significant demand to develop an efficient and open-source numerical tool for the investigation of such phenomena. Because of the nonlinear, time-dependent, and multiphysical nature of these various multiphase FSI problems, a simulation tool that is both robust and highly scalable (in parallel computing terms) is challenging. There are notable commercial FSI solvers. However, few of them can achieve both numerical robustness and high scalability while also being able to tackle multiphase FSI problems. Commercial software, such as ANSYS (Rao, 2003) and COMSOL (Curtis et al., 2013), provide fully coupled FSI simulations. Compared with commercial software, open-source codes have advantages in removing parallel scaling-related costs (among other benefits such as enabling the implementation of bespoke solvers or algorithms, as well as transparency).

Martínez-Ferrer et al. (2018) established an FSI simulation tool using OpenFOAM. Both fluid and structure domains were discretised with the finite volume method and solved with OpenFOAM using up to four CPU cores. Integrating both fluid and structural domains in a single library with a code-specific data mapping method is a good way to model multiphysics problems with two or three computational domains. However, in the long run, maintaining these codes and gradually adding more computational domains will be onerous work. It is also less flexible in dealing with problems that exceed the ability of any individual established code. Performance of the overall FSI solution is also limited by the scalability of the host codebase, whereas a partitioned approach allows individual components with the system to be scaled according to their problem size, allowing for more optimal use of computing resources.

In this study, we aim to establish a new parallel partitioned multiphase FSI simulation framework using open-source codes. We adopt a partitioned approach, ensuring good use is made of existing open-source software while allowing design flexibility and reduced future maintenance efforts. For a partitioned approach, a stable and accurate coupling algorithm with good scalability and flexibility is required. Several key coupling libraries provide algorithms for FSI simulations, such as Comana (König et al., 2016), OpenFPCI (Hewitt et al., 2019), preCICE (Bungartz et al., 2016), and the multiscale universal interface (MUI) coupling library (Tang et al., 2015). The Comana code is not open source, and OpenFPCI is designed as a coupling framework between a specific structural code, ParaFEM (Smith et al., 2007), and OpenFOAM fluid solver. OpenFOAM also acts as a host in OpenFPCI. In this work, we employ the MUI library as the interface coupling tool between fluid and structure domains. It provides highly flexible domain couplings, as it allows an arbitrary number of codes to communicate with one another via the message passing interface (MPI) communications using a cloud of data points (rather than a mesh), combined with high-order interpolation schemes to facilitate data use between dissimilar methods or discretisation. MUI coupling utilities (CU) with FSI coupling algorithms have been developed to achieve a tight and stable coupling. OpenFOAM (Weller et al., 1998) and FEniCS (Alnæs et al., 2015) are adopted as the multiphase computational fluid dynamics (CFD) and computational structure mechanics (CSM) solvers, respectively. All three codes (i.e., MUI, FEniCS, and OpenFOAM) are equally important in this framework. Codes used for this multiphase FSI

Received August 5, 2021; updated and further revised manuscript received by the editors September 27, 2021. The original version (prior to the final updated and revised manuscript) was presented at the Thirty-first International Ocean and Polar Engineering Conference (ISOPE-2021), Rhodes, Greece (virtual), June 20–25, 2021.

KEY WORDS: Multiphase simulation, partitioned fluid-structure interaction, multiphysics-coupled modelling.