preCICE for OpenFOAM: from CHT and FSI to a general-purpose plug-in

Gerasimos Chourdakis*, **Benjamin Uekermann***, **Derek Risseeuw**†, **Hans-Joachim Bungartz**†

* Department of Informatics
Technical University of Munich
Informatik 5, Boltzmannstr. 3, 85748 Garching b. Mnchen, Germany
e-mail: chourdak@in.tum.de, web page: https://www5.in.tum.de/

† Delft University of Technology
Kluyverweg 1, P.O. Box 5058, 2629HS Delft, The Netherlands

ABSTRACT

Designing a general-purpose coupling tool targeting several users from different fields poses different challenges than coupling two simulation codes with a specific research question in mind. The preCICE coupling library[1] is a problem-agnostic tool with a high-level, easy to use programming interface, which hides the complexity of the coupling. This design accelerates the development tasks of the curious researcher and lets her spend more time focused on the challenges of the problem at hand. But what if the researcher could study an arbitrary question without writing any additional line of code, just by using the tools she already knows?

preCICE comes with a wide range of ready-to-use adapters[2]. Some of them modify a specific solver of a software package (e.g. the SU2 or CalculiX adapters), while other are meant to be examples for an adapted code (e.g. the FEniCS or deal-ii adapters), depending on the flexibility and the use cases of the software package at hand. The official OpenFOAM adapter[3] goes one step further, taking the form of an OpenFOAM function object that does not require any code changes and works with arbitrary solvers and a wide range of OpenFOAM versions.

Having started with support only for conjugate heat transfer[4], it was recently extended to support moving mesh solvers for fluid-structure interaction[5]. The solver-agnostic design led to a small amount of required code changes. With an extended set of features and supported OpenFOAM versions at hand, the code was later generalized even further, preparing for the addition of a fluid-fluid coupling module.

This talk will review the design decisions and present the challenges of covering every targeted problem type with the same, solver-agnostic adapter for OpenFOAM.

REFERENCES

[1] Bungartz, H.-J. et al. *preCICE – A Fully Parallel Library for Multi-Physics Surface Coupling*. Computers and Fluids (2016), Volume 141, p. 250–258.

[2] Uekermann, B. et al. *Official preCICE Adapters for Standard Open-Source Solvers*. Proceedings of the 7th GACM Colloquium on Computational Mechanics for Young Scientists from Academia (2017).

[3] Chourdakis, G. *A general OpenFOAM adapter for the coupling library preCICE*. Master's thesis (2017), Department of Informatics, Technical University of Munich.

[4] Cheung Yau, L. *Conjugate heat transfer with the multiphysics coupling library preCICE*. Master's thesis (2016), Department of Informatics, Technical University of Munich.

[5] Code repository of the OpenFOAM adapter on GitHub, Pull Request #56 *Add support for Fluid-Structure Interaction* https://github.com/precice/openfoam-adapter