CAN MY OPENFOAM SOLVER EASILY BE COUPLED WITH PRECICE?

Gerasimos Chourdakis[*,1], Benjamin Uekermann[2] and Hans-Joachim Bungartz[3]

[1,2,3] Technical University of Munich, Boltzmannstraße 3, D-85748 Garching bei München, [1] gerasimos.chourdakis@tum.de [2] uekerman@in.tum.de [3] bungartz@in.tum.de

Keywords: *Partitioned Approach, Multiphysics Problems, Conjugate Heat Transfer, Fluid-Structure Interaction*

Most probably, yes! The new, official preCICE adapter [1, 2, 3] for OpenFOAM [4] supports a wide range of standard and in-house solvers, without requiring any source code modifications. Having been developed as a function object, this adapter can be loaded at runtime, minimizing the friction for the user. Separating the core functionality from the problem-specific boundary conditions, it is also easily extensible, enabling OpenFOAM to work with other simulation software in complex multi-physics simulations. This talk will present how to solve typical multi-physics problems with OpenFOAM and preCICE. Furthermore, it will give insight on how to extend the adapter to couple additional variables.

REFERENCES

[1] Hans-Joachim Bungartz et al. "preCICE – A fully parallel library for multi-physics surface coupling". In: *Comput. & Fluids* 141, pp. 250–258, Elsevier, 2016.

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

[3] Gerasimos Chourdakis "A general OpenFOAM adapter for the coupling library preCICE". Master's thesis, Institut für Informatik, Technische Universität München, 2017.

[4] Henry Weller et al. "A Tensorial Approach to Computational Continuum Mechanics Using Object-oriented Techniques". In: *Comput. Phys.* 12 (6), pp. 620–631, 1998.