Chair of Scientific Computing
Department of Informatics
Technical University of Munich

Scalable coupled simulations with OpenFOAM® and other solvers

Setting up Multi-Physics simulations like playing LEGO®, without stepping on the pieces

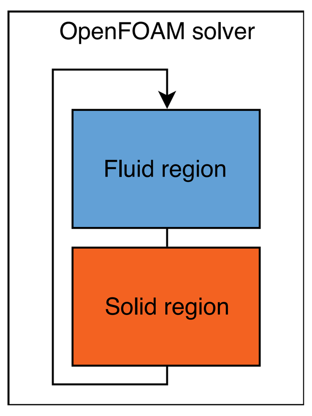
Hans-Joachim Bungartz[1], **Gerasimos Chourdakis**[1], **Derek Risseeuw**[2], **Alexander Rusch**[1], **Benjamin Uekermann**[1]
[1]{bungartz, chourdak, uekerman}@in.tum.de and alexander.rusch@tum.de, Scientific Computing, Technical University of Munich
[2]d.risseeuw@student.tudelft.nl, Faculty of Aerospace Engineering, Delft University of Technology

Multi-Physics in OpenFOAM

Approaches and problems

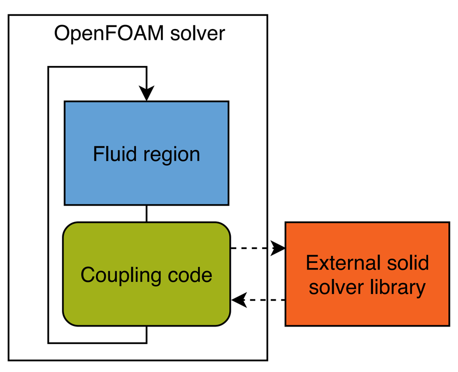
Framework approach: Two different regions in an OpenFOAM solver, solved sequentially. Examples: chtMultiRegionFoam (intrinsic), fsiFoam (only in foam-extend) [1].

+ Everything inside OpenFOAM
– Limited to specific, OpenFOAM-only solvers
– Limited coupling numerics
– Limited scalability
– Partitioned numerics, but monolithic software



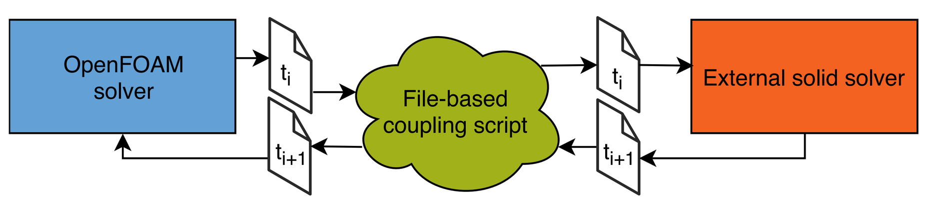
Master - Slave approach: The OpenFOAM solver calls an external solver library. Example: OpenFPCI [2].

+ Not limited to OpenFOAM solvers
– Invasive (convert solver to library)
– Difficult extension to > 2 solvers
– Limited scalability
– Tight setup, difficult to maintain



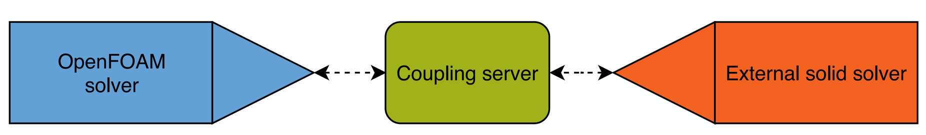
Files-based approach: Every solver works independently, solving every coupling timestep as a complete simulation. An external script (often in-house) processes their results files and adjusts their configuration files, restarting them. Example: Galanin et. al. [3].

+ Non-invasive
+ Intuitive and easy to debug
– Very slow



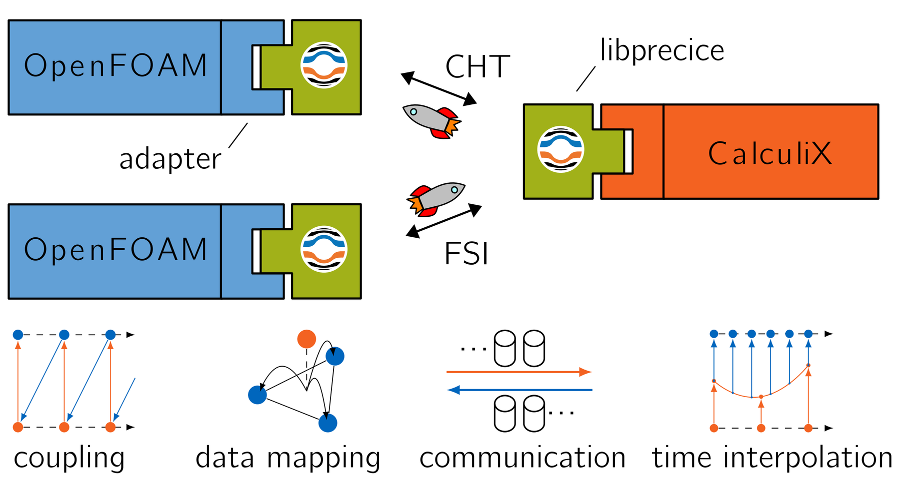
Server library approach: Every solver calls an external library, which couples them. The solvers do not communicate directly. Examples: MpCCI (commercial) [4], EMPIRE [5], ifls [6].

+ Flexible
+ Advanced coupling numerics
– Limited scalability
– Additional executable to handle



Solution: preCICE coupling library

preCICE (www.precice.org) [7] follows a completely parallel, peer-to-peer approach [8]. preCICE treats each solver as a "black-box" and handles all the functionality required for the coupling. Each participating solver loads the preCICE library through an "adapter" [9].



preCICE supports (parallel) implicit coupling with Quasi-Newton acceleration, RBF mapping, and communication via MPI or TCP/IP sockets. Time interpolation schemes are under development.

OpenFOAM adapter for preCICE [10]

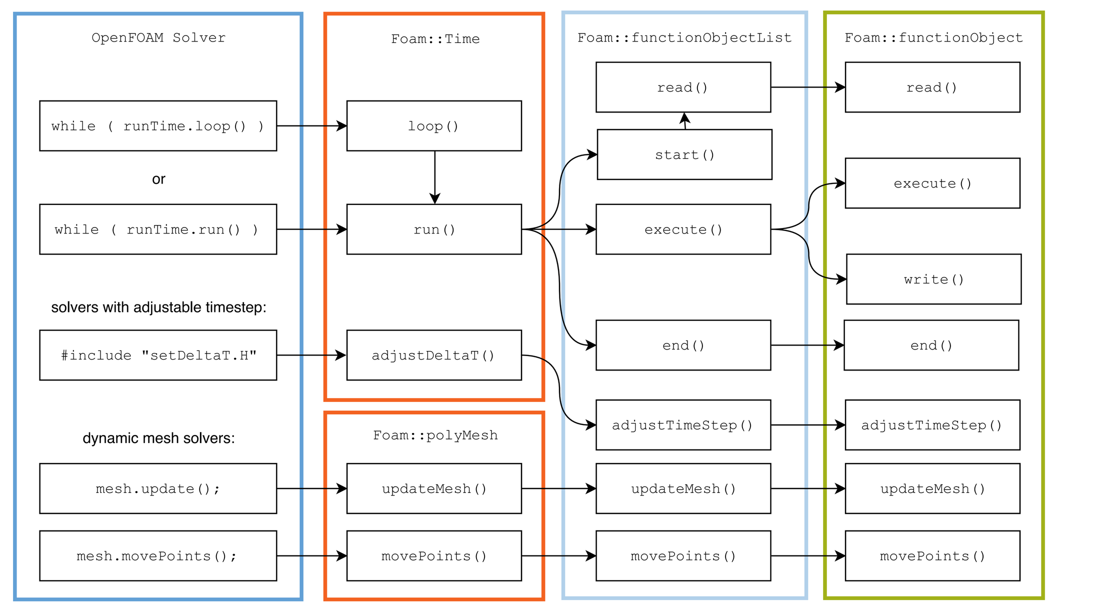
One adapter, no changes in the solver

Previous approach: Modify the solver to add calls to preCICE → solver-specific. Examples: FOAM-FSI [11], L. Cheung [12], K. Rave [13], D. Schneider [14].

OpenFOAM function objects: Inject code at specific, pre-defined points, without modifying the solver's code or re-compiling.

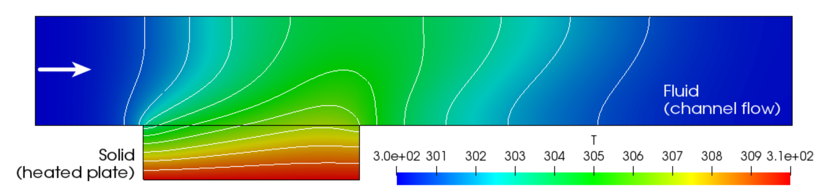
Compatible: Designed for current versions of openfoam.com and openfoam.org.

Challenges: Access everything through the objects' registry and keep the code general.

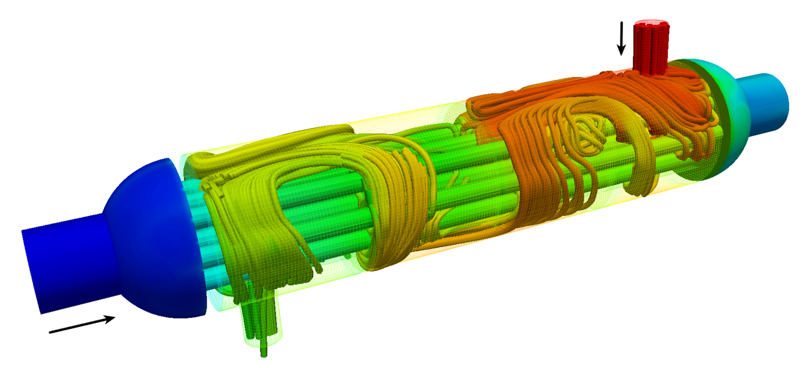


Results & Tutorials

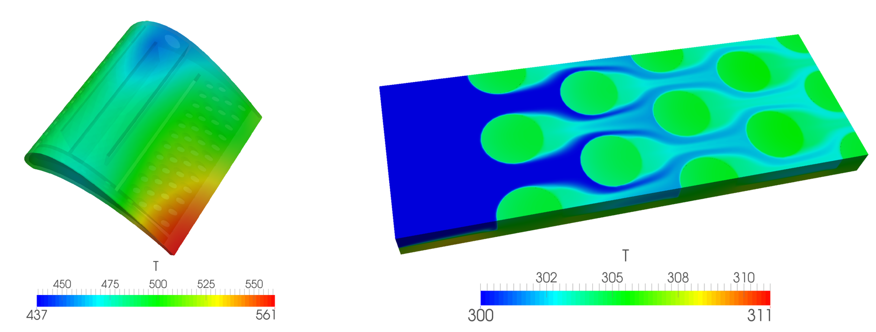
Flow over a heat plate: CHT tutorial, validated.



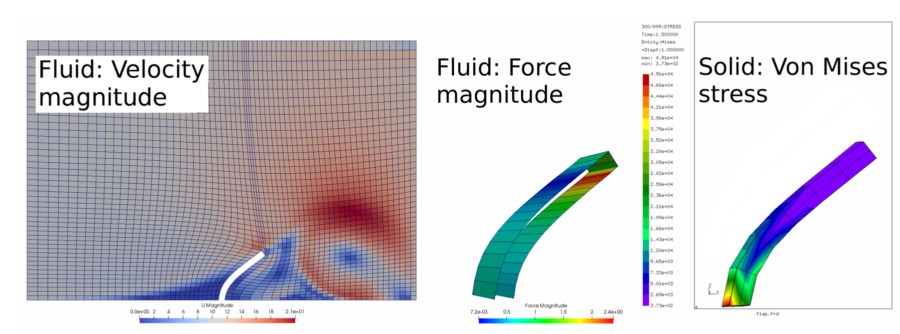
Shell-and-tubes heat exchanger: Multi-coupling CHT tutorial with OpenFOAM and CalculiX (calculix.de). Compared to chtMultiRegionFoam, a performance benefit has been observed for this scenario. A rigorous performance study is future work.



More CHT scenarios: Turbine blade cooling and pin-fin cooling (simulations by L. Cheung [12])

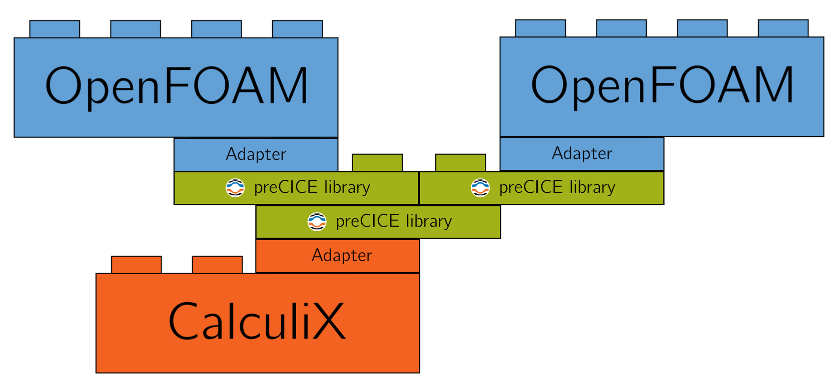


Initial FSI results: Flap perpendicular to the flow with OpenFOAM (left, center) and CalculiX (right)



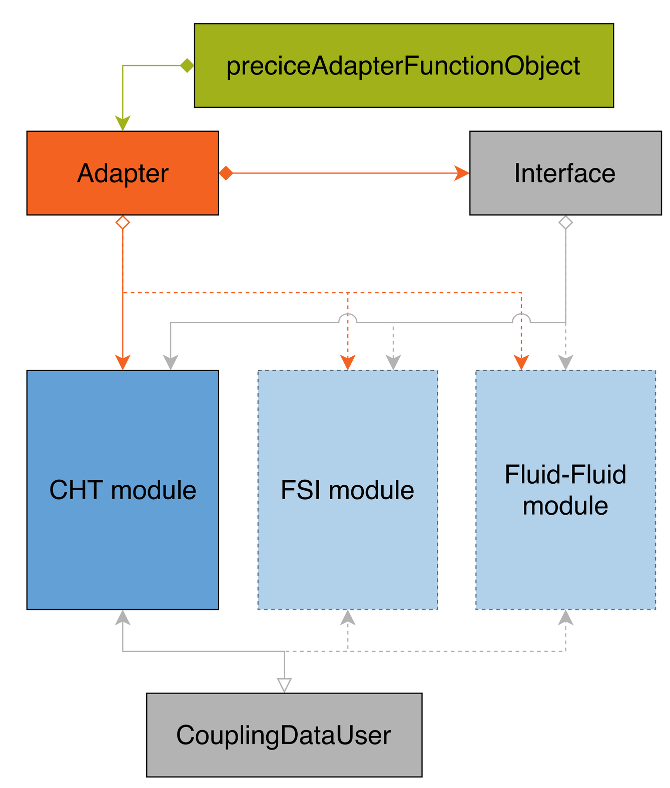
Easy to use

+ **Load at runtime:** Add 3 lines in your `system/controlDict`
+ **Easy to configure:** Small (approx. 7 lines) YAML file (plan to replace it with an OpenFOAM dictionary)
+ **No matter which partner:** The adapter only knows about preCICE.



Next steps

The adapter can be extended to support other problem types, such as Fluid-Fluid multi-model coupling.



Acknowledgments: preCICE is currently being developed at the Technical University of Munich by Hans-Joachim Bungartz, Gerasimos Chourdakis, Benjamin Rüth, Alexander Rusch, Frédéric Simonis, and Benjamin Uekermann and at the University of Stuttgart by Florian Lindner, Miriam Mehl, and Amin Totounferoush. preCICE is funded by SPPEXA, the DFG Priority Programme 1648 – Software for Exascale Computing (www.sppexa.de) and DFG's initiative to support sustainable research software. The work of Gerasimos Chourdakis was supported by a DAAD scholarship (www.daad.de).

References:

[1] Tukovic, Z., Cardiff, P., Karac, A., Jasak, H., Ivankovic, A. (2014) OpenFOAM Library for Fluid Structure Interaction. 9th OpenFOAM Workshop.
[2] Hewitt, S., Margetts L., and Revell, A. (2017). Parallel Performance of an Open Source Fluid Structure Interaction Application. In 25th UKACM Conference on Computational Mechanics.
[3] Galanin, M. P., Zhukov, V. T., Klyushnev, N. V., Kuzmina, K. S., Lukin, V. V., Marchevsky, I. K., and Rodin, A. S. (2018). Implementation of an iterative algorithm for the coupled heat transfer in case of high-speed flow around a body. In Computers & Fluids.
[4] Wirth, N., Bayrasy, P., Landvogt, B., Wolf, K., Cecutti, F. and Lewandowski, T. (2017). Analysis and Optimization of Flow Around Flexible Wings and Blades Using the Standard Co-simulation Interface MpCCI. In Recent Progress in Flow Control for Practical Flows, 283-321. Springer.
[5] Sicklinger, S.A., Wang, T., Lerch, C., Wüchner, R. and Bletzinger, K.U. (2012). EMPIRE: A N-Code Coupling Tool for Multiphysic Co-Simulations with OpenFOAM. In 7th OpenFOAM Workshop.
[6] Müller, M., Haupt, M., and Horst, P. (2018). Socket-Based Coupling of OpenFOAM and Abaqus to Simulate Vertical Water Entry of Rigid and Deformable Structures. In 6th ESI OpenFOAM User Conference.
[7] Bungartz, H.-J., Lindner, F., Gatzhammer, B., Mehl, M., Scheufele, K., Shukaev, A., and Uekermann, B. (2016). preCICE – A fully parallel library for multi-physics surface coupling. In Comput. & Fluids, 141(Supplement C), 250–258.
[8] Uekermann, B. (2016). Partitioned Fluid-Structure Interaction on Massively Parallel Systems. PhD Thesis. Technical University of Munich.
[9] Uekermann, B., Bungartz, H.-J., Cheung Yau, L., Chourdakis, G. and Rusch, A. (2017). Official preCICE Adapters for Standard Open-Source Solvers. In 7th GACM Colloquium on Computational Mechanics for Young Scientists from Academia.
[10] Chourdakis, G. (2017). A general OpenFOAM adapter for the coupling library preCICE. Master's Thesis. Technical University of Munich.
[11] Blom D. (2016). FOAM-FSI: Fluid-Structure Interaction solvers for foam-extend.
[12] Cheung Yau, L. (2016). Conjugate Heat Transfer with the Multiphysics Coupling Library preCICE. Master's Thesis. Techn. Univ. of Munich.
[13] Rave, K. (2017). Kopplung von OpenFOAM und deal.II Gleichungslösern mit preCICE zur Simulation multiphysikalischer Probleme. Master's thesis (unpublished), Lehrstuhl für Strömungsmechanik, Universität Siegen.
[14] Schneider D. (2018). Simulation von Fluid-Struktur-Interaktion mit der Kopplungsbibliothek preCICE. Bachelor's thesis (unpublished), Lehrstuhl für Strömungsmechanik, Universität Siegen.

www.precice.org
github.com/precice
chourdak@in.tum.de