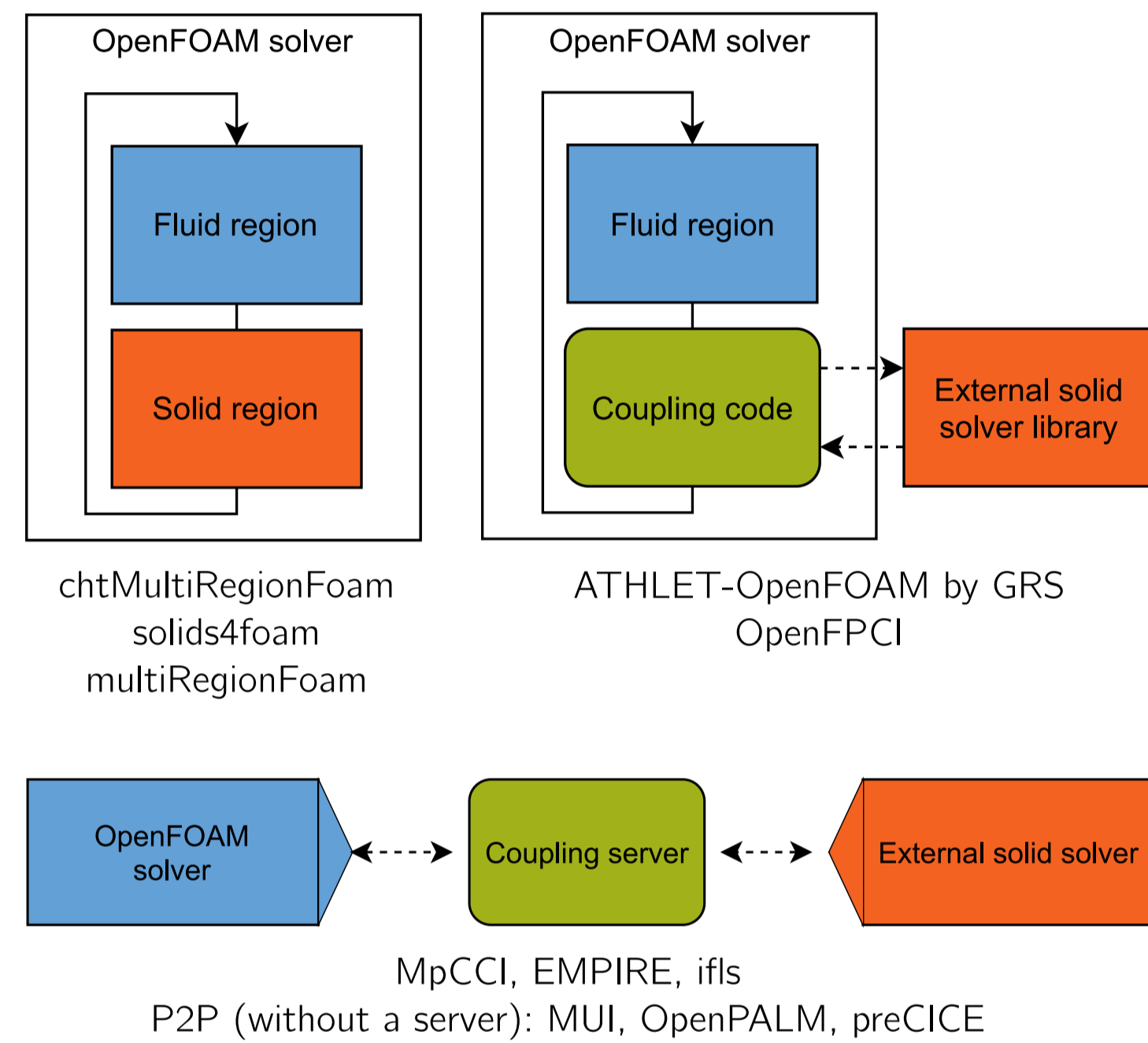


Plain language summary

Users of simulation codes often wish to extend simulations to include phenomena solved in OpenFOAM. Such problems appear in cooling devices, on flexible wings, or even in nuclear fusion reactors. The OpenFOAM-preCICE adapter allows users to quickly couple OpenFOAM to arbitrary other codes, bridging their communities, and opening new application possibilities for OpenFOAM.

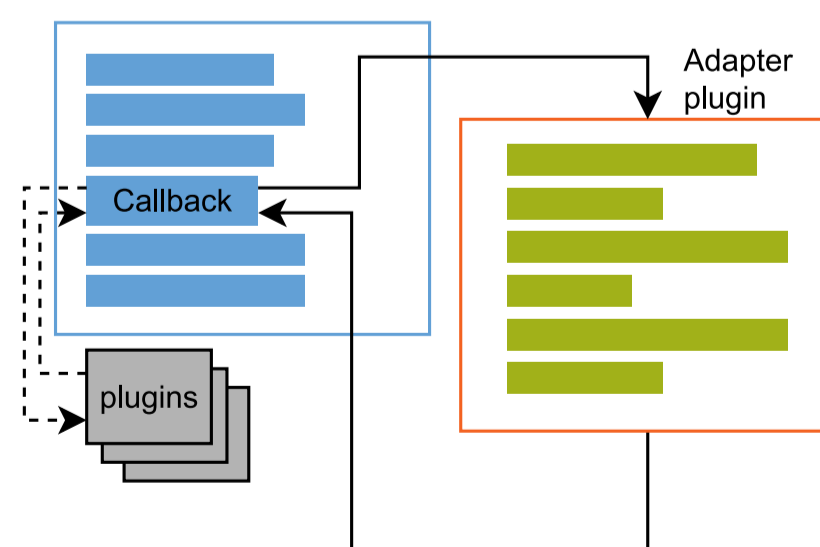
Common coupling approaches



The OpenFOAM-preCICE adapter

A function object that calls preCICE

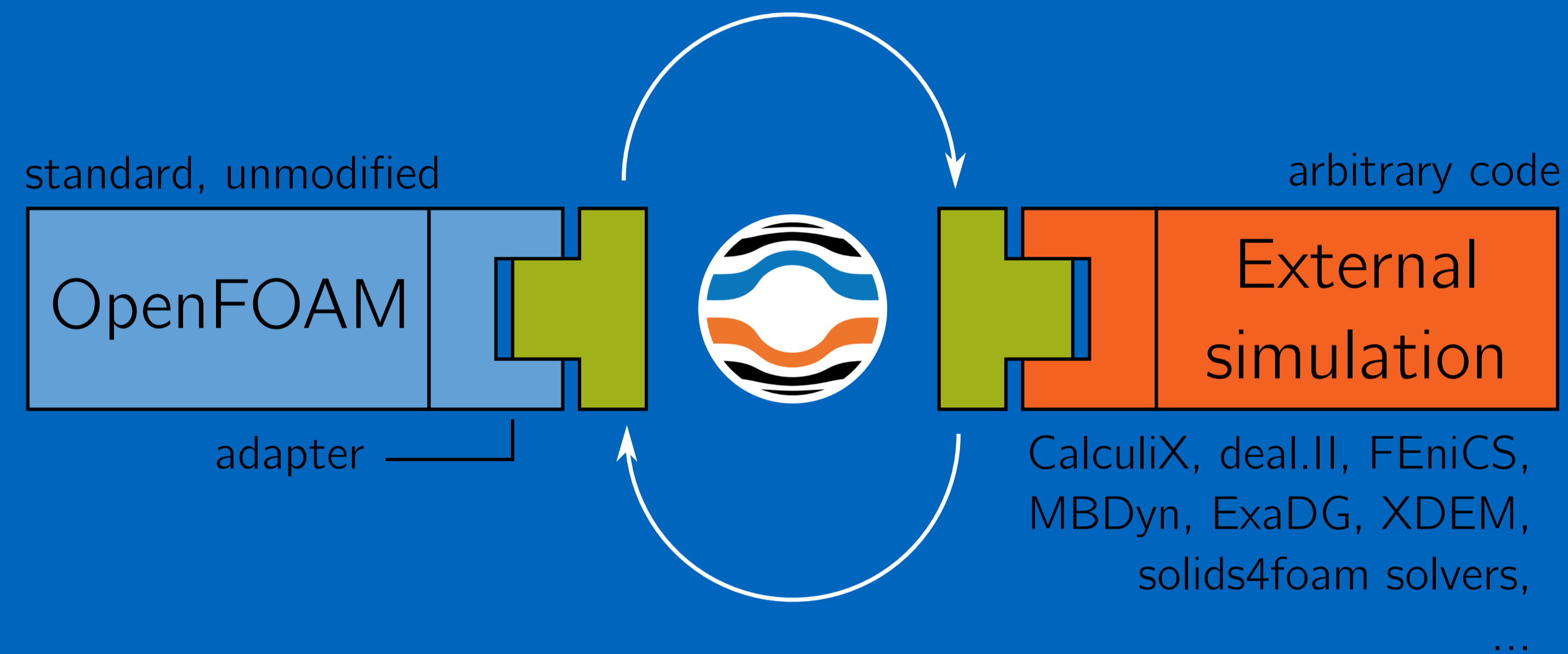
- Works with standard OpenFOAM solvers
- OpenFOAM v2306, OpenFOAM 10, ...
- On github.com/precice/openfoam-adapter



Extensions and community

- Coming up: volume coupling, fluid-fluid
- Several third-party application papers
- Active community forum

Couple OpenFOAM^[1] with any external simulation code via the coupling library preCICE^[2]



Gerasimos Chourdakis*, David Schneider, Benjamin Uekermann

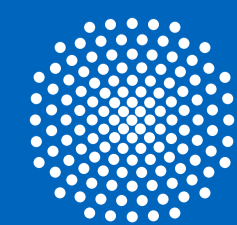
Coupling OpenFOAM with external simulation codes using OpenFOAM-preCICE

gerasimos.chourdakis@tum.de

SCCS, TUM School of Computation, Information and Technology

All figures originate from the OpenFOAM-preCICE paper [1].

- [1] G.Chourdakis, D. Schneider, B. Uekermann (2023), OpenFOAM-preCICE: Coupling OpenFOAM with external solvers for multi-physics simulations OpenFOAM@ Journal, 3:1-25, <https://doi.org/10.51560/ofj.v3.88>
- [2] G. Chourdakis, K. Davis, B. Rodenberg, M. Schulte, F. Simonis, B. Uekermann, et al. (2022), preCICE v2: A sustainable and user-friendly coupling library [version 2; peer review: 2 approved] Open Res Europe, 2:51, <https://doi.org/10.12688/openreseurope.14445.2>

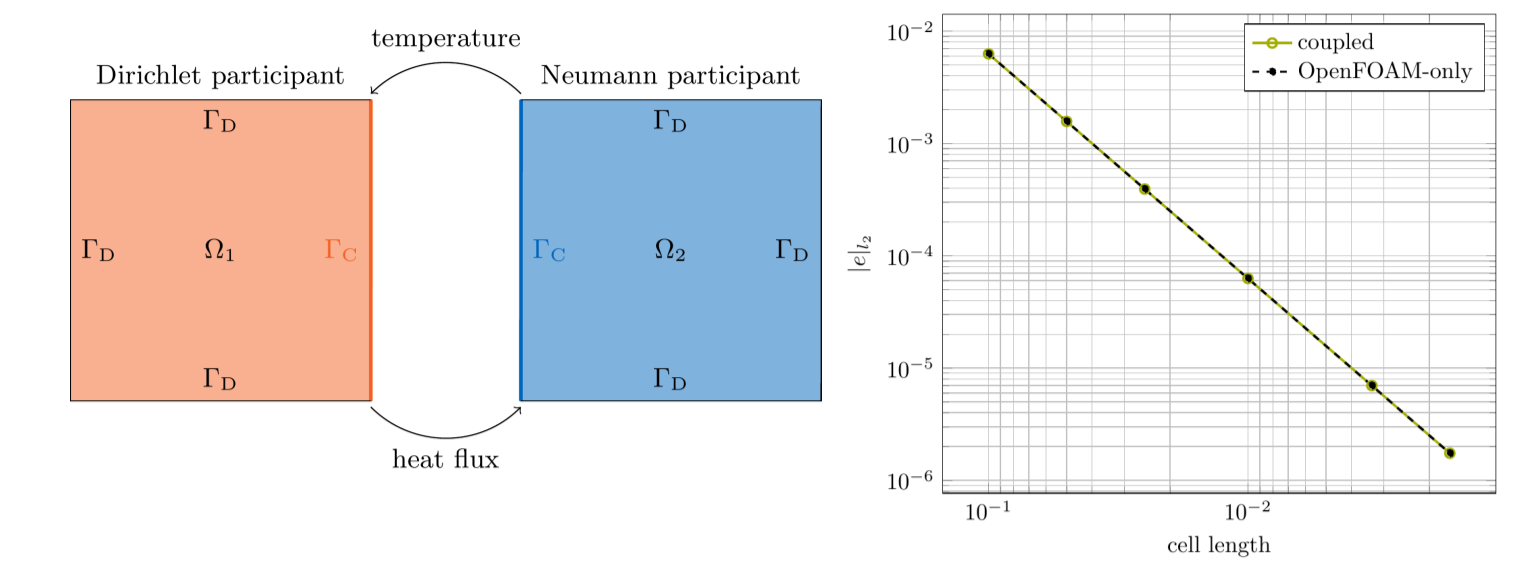


University of Stuttgart
Germany



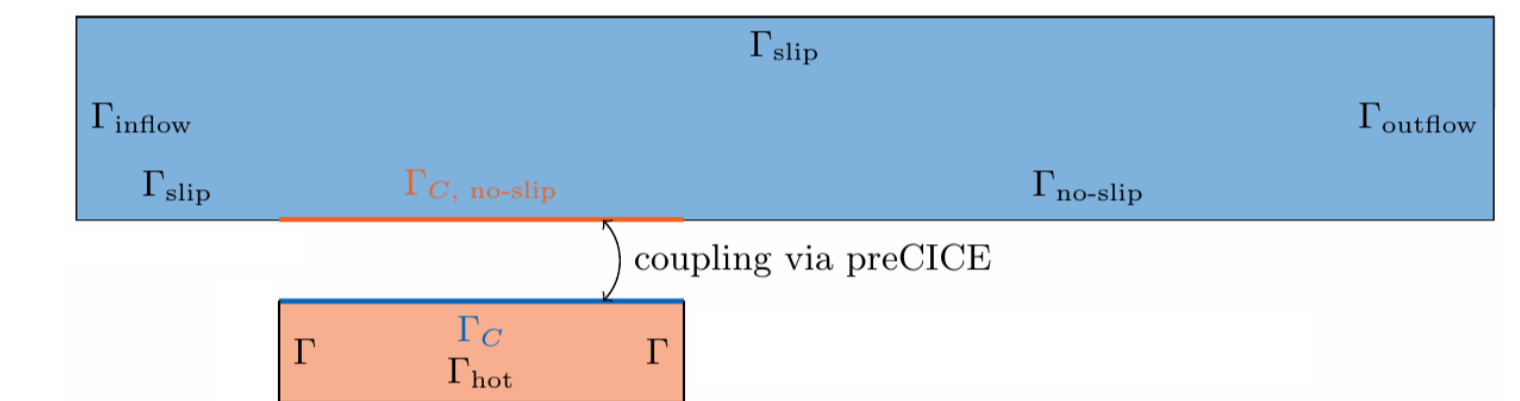
Partitioned heat transfer

- 2x mod. laplacianFoam, Aitken, NN map
- perfect comparison to 1x laplacianFoam



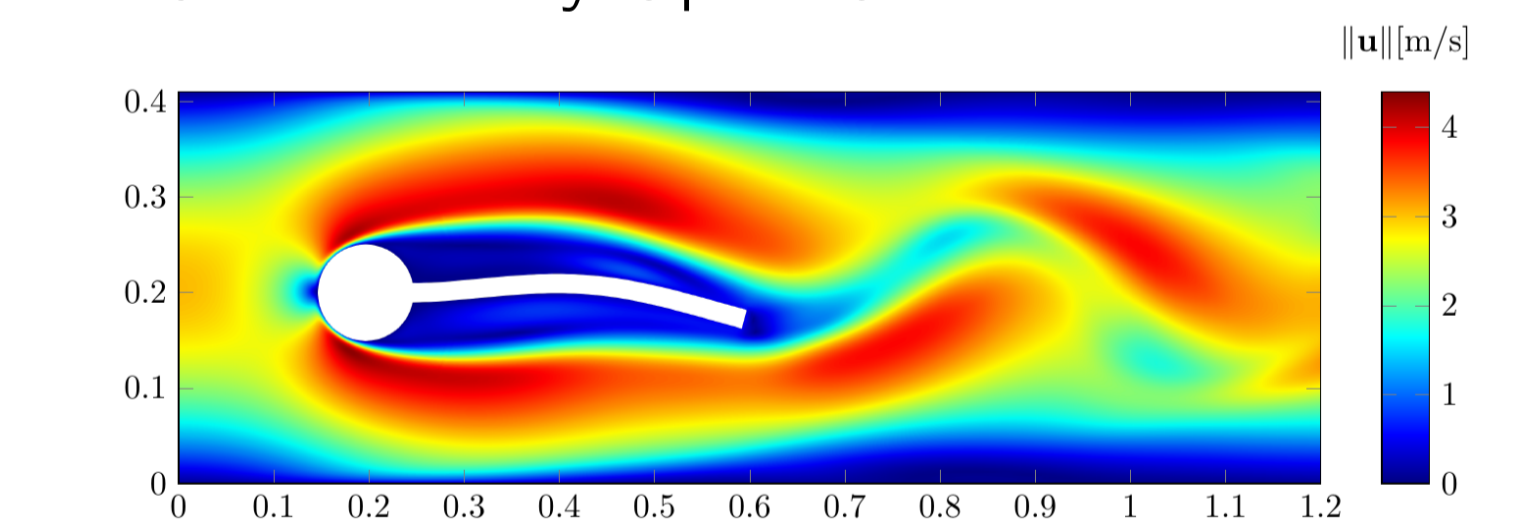
Conjugate heat transfer

- Setup by Vynnycky et al. (1998)
- buoyantPimpleFoam + laplacianFoam
- steady-state and transient, explicit coupling
- perfect comparison to chtMultiRegionFoam



2D fluid-structure interaction

- Setup by Turek & Hron (2006)
- pimpleFoam + ExaDG, IQN, RBF mapping
- FSI3 almost identical to reference
- FSI2 limited by OpenFOAM mesh motion



3D fluid-structure interaction

- Setup by Schott, Ager, Wall (2019)
- pimpleFoam + deal.II, IQN, RBF mapping
- Results very close to reference
- Runtime: Adapter takes 1-2% of total

