The preCICE coupling library in 3min
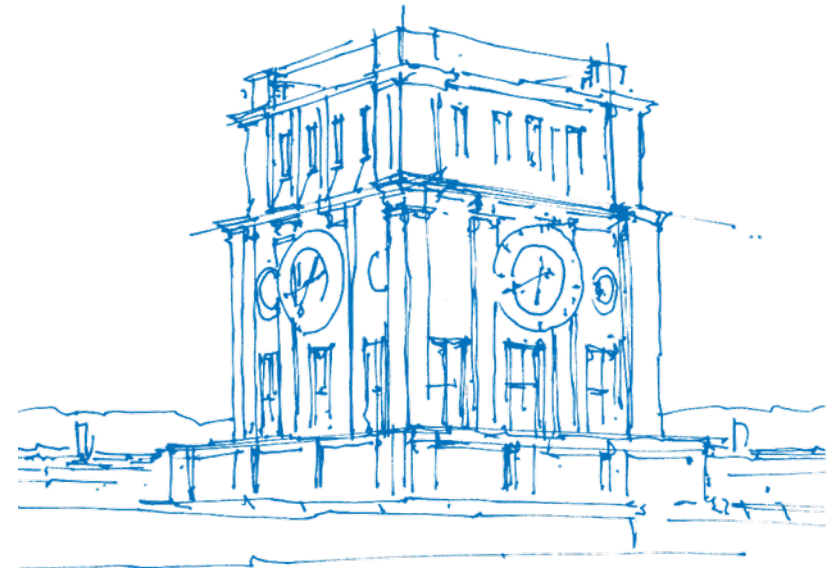
Gerasimos Chourdakis

Technical University of Munich

Department of Informatics

Scientific Computing in Computer Science

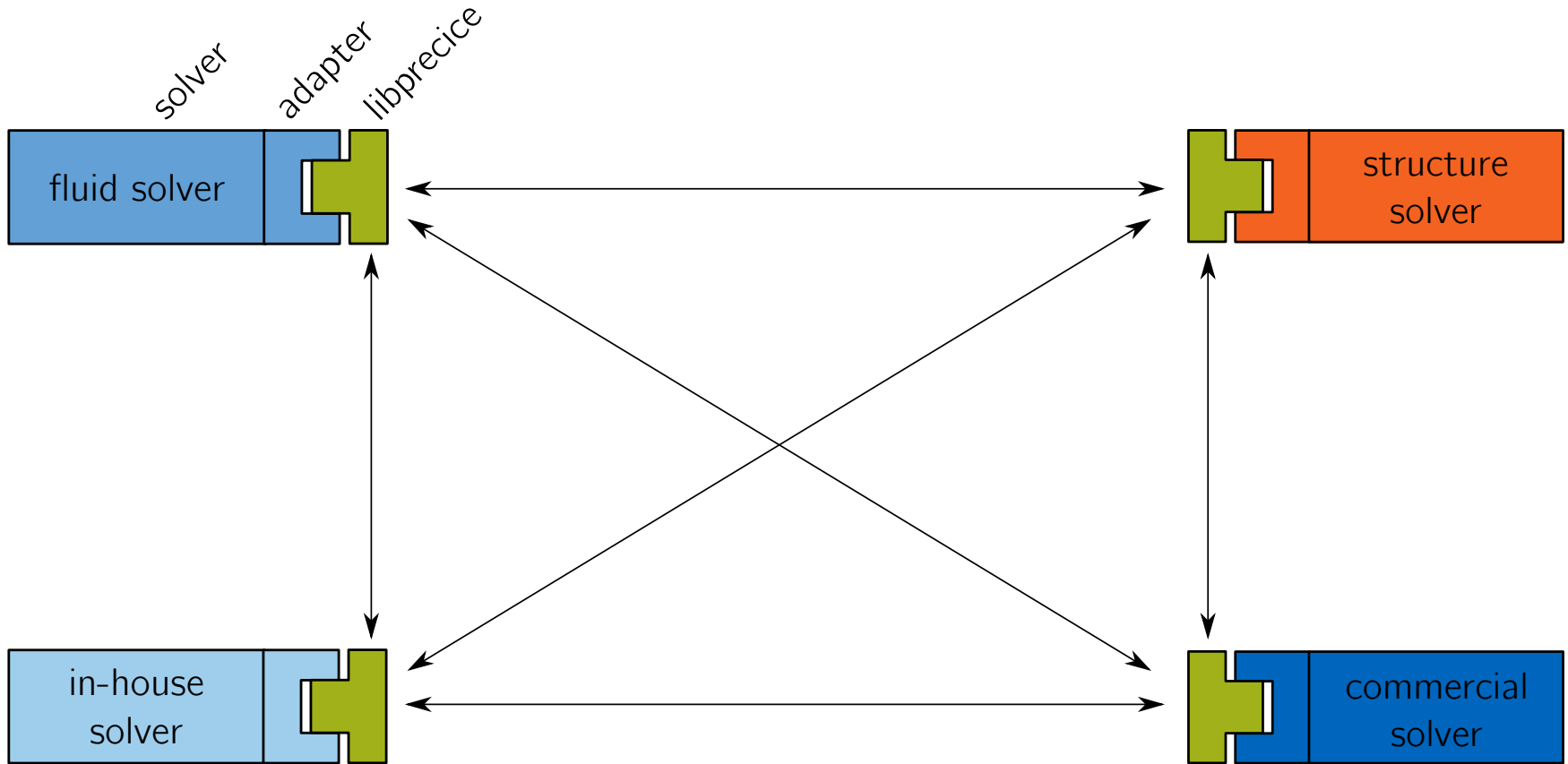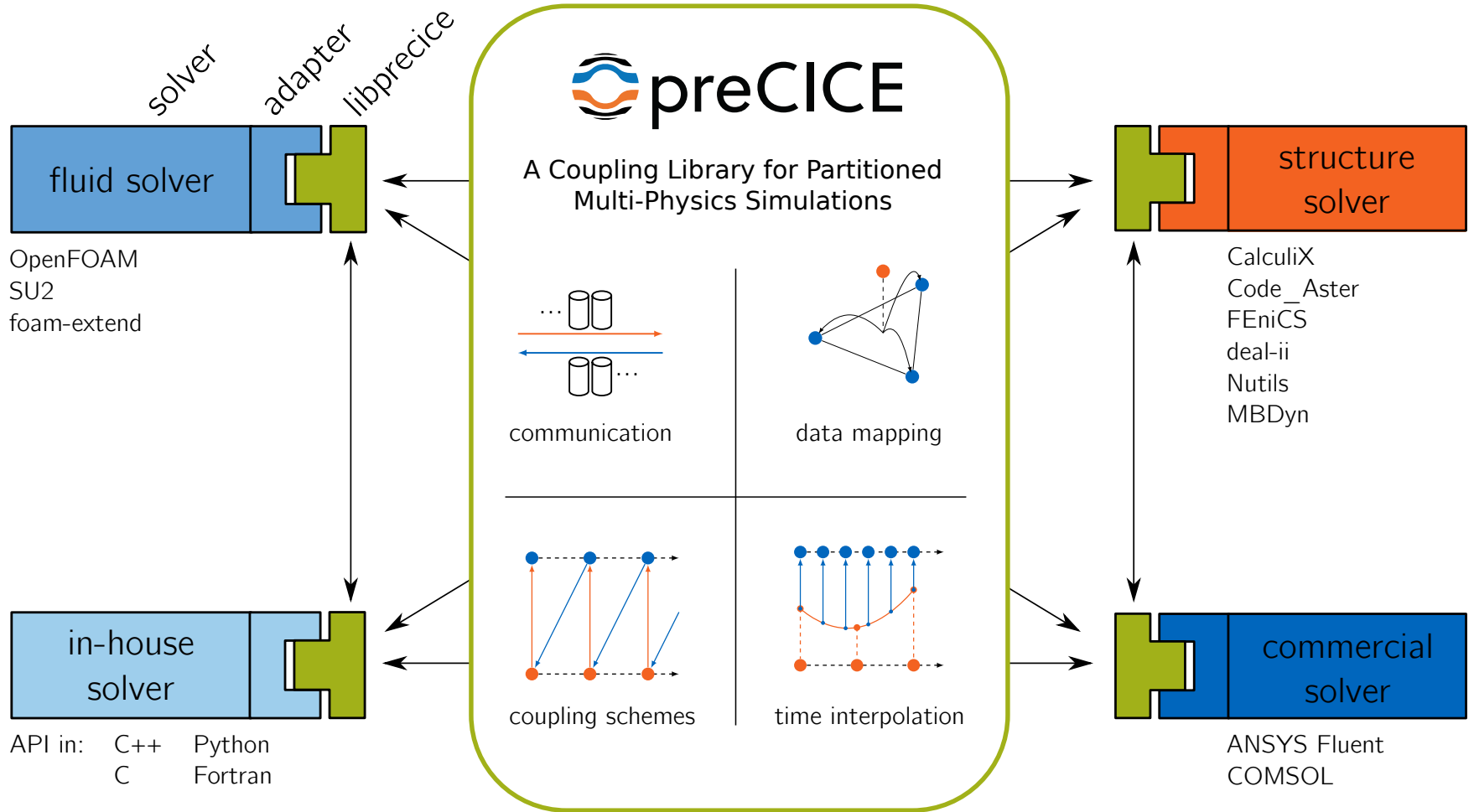14th OpenFOAM Workshop - Duisburg, Germany

July 24, 2019

What is preCICE?
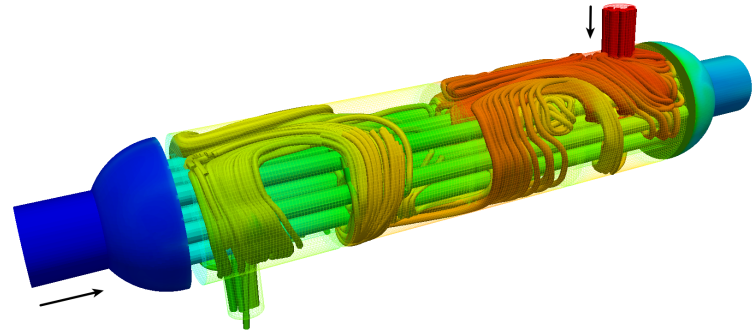
What is preCICE?

What is preCICE?

What is preCICE?

Get started

Tutorials on `www.precice.org/resources` (step-by-step):
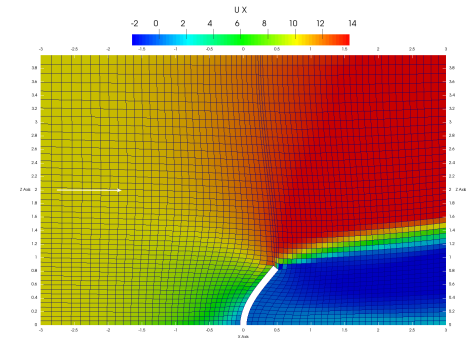
CHT: Shell-and-Tubes Heat Exchanger

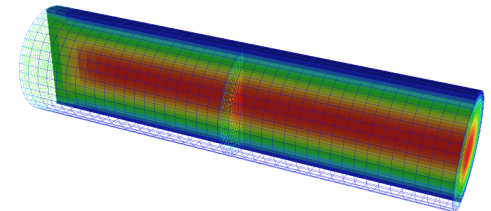- buoyantSimpleFoam (x2) + CalculiX
- Multiple participants

FSI: Flow in a channel with a perpendicular flap

- {pimpleFoam + CalculiX}
  or {pimpleFoam + deal.ii} or {SU2 + CalculiX}
- Also on your browser: `run.precice.org`

FF: Flow in a partitioned pipe (soon)

- pimpleFoam (x2) or sonicLiquidFoam (x2)
- Currently testing, only Readme

Community

Metrics: 10+ years, 45k lines of C++11, 120+ stars on GitHub, 30+ user groups

Everything is on GitHub! (recommended: code reviews)

Some nice issues to start from:

- Supporting multiple OpenFOAM versions (OpenFOAM adapter issue #32)
  - ...with the same source code
- Converting to an OpenFOAM dictionary (OpenFOAM adapter issue #30)
  - ...while keeping the same logic (list of dictionaries)

Software sustainability: project preDOM (building, testing, distributing, ...)

Funding

preCICE is free because of the support of:

H2020 grant 754462

And the code/issues/testing/documentation contributions of people like you (thank you!).

Summary

preCICE + Open∇FOAM / OpenFOAM (The OpenFOAM Foundation)

preCICE:	coupling library for partitioned multi-physics simulations
Communication:	MPI, TCP/IP sockets
Mapping:	RBF, nearest-projection, nearest-neighbor
Coupling:	explicit/implicit (= iterative), Interface Quasi-Newton acceleration
OpenFOAM adapter:	CHT, FSI, fluid-fluid coupling

Summary

preCICE + Open∇FOAM

OpenFOAM
The OpenFOAM Foundation

preCICE:	coupling library for partitioned multi-physics simulations
Communication:	MPI, TCP/IP sockets
Mapping:	RBF, nearest-projection, nearest-neighbor
Coupling:	explicit/implicit (= iterative), Interface Quasi-Newton acceleration
OpenFOAM adapter:	CHT, FSI, fluid-fluid coupling

🌐 `www.precice.org`

🐙 `github.com/precice`

🐦 `@preCICE_org, @_makCh`

🏛 `www5.in.tum.de/~chourdak`

@ `chourdak@in.tum.de`

Summary

preCICE + Open▽FOAM

OpenFOAM
The OpenFOAM Foundation

preCICE: coupling library for partitioned multi-physics simulations

Communication: MPI, TCP/IP sockets

Mapping: RBF, nearest-projection, nearest-neighbor

Coupling: explicit/implicit (= iterative), Interface Quasi-Newton acceleration

OpenFOAM adapter: CHT, FSI, fluid-fluid coupling

- www.precice.org
- github.com/precice
- @preCICE_org, @_makCh
- www5.in.tum.de/~chourdak
- @ chourdak@in.tum.de

Preview of doughnuts that will be served in the preCICE Workshop 2020 (February 17-18, Munich).