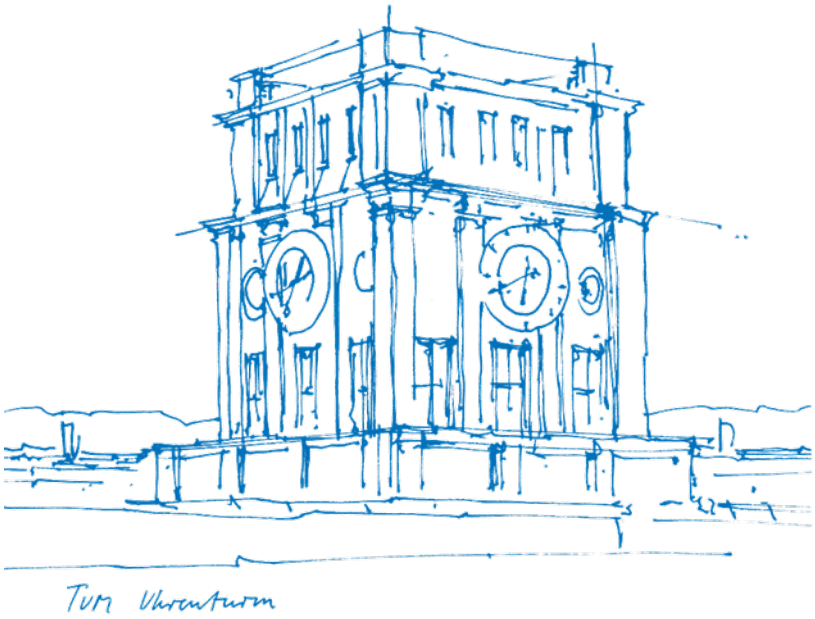
Coupling OpenFOAM to different solvers, physics, models, and dimensions using preCICE

Gerasimos Chourdakis[1], Benjamin Uekermann[2]

[1]Technical University of Munich
Department of Informatics
Scientific Computing in Computer Science

[2] Eindhoven University of Technology
Department of Mechanical Engineering
Energy Technology

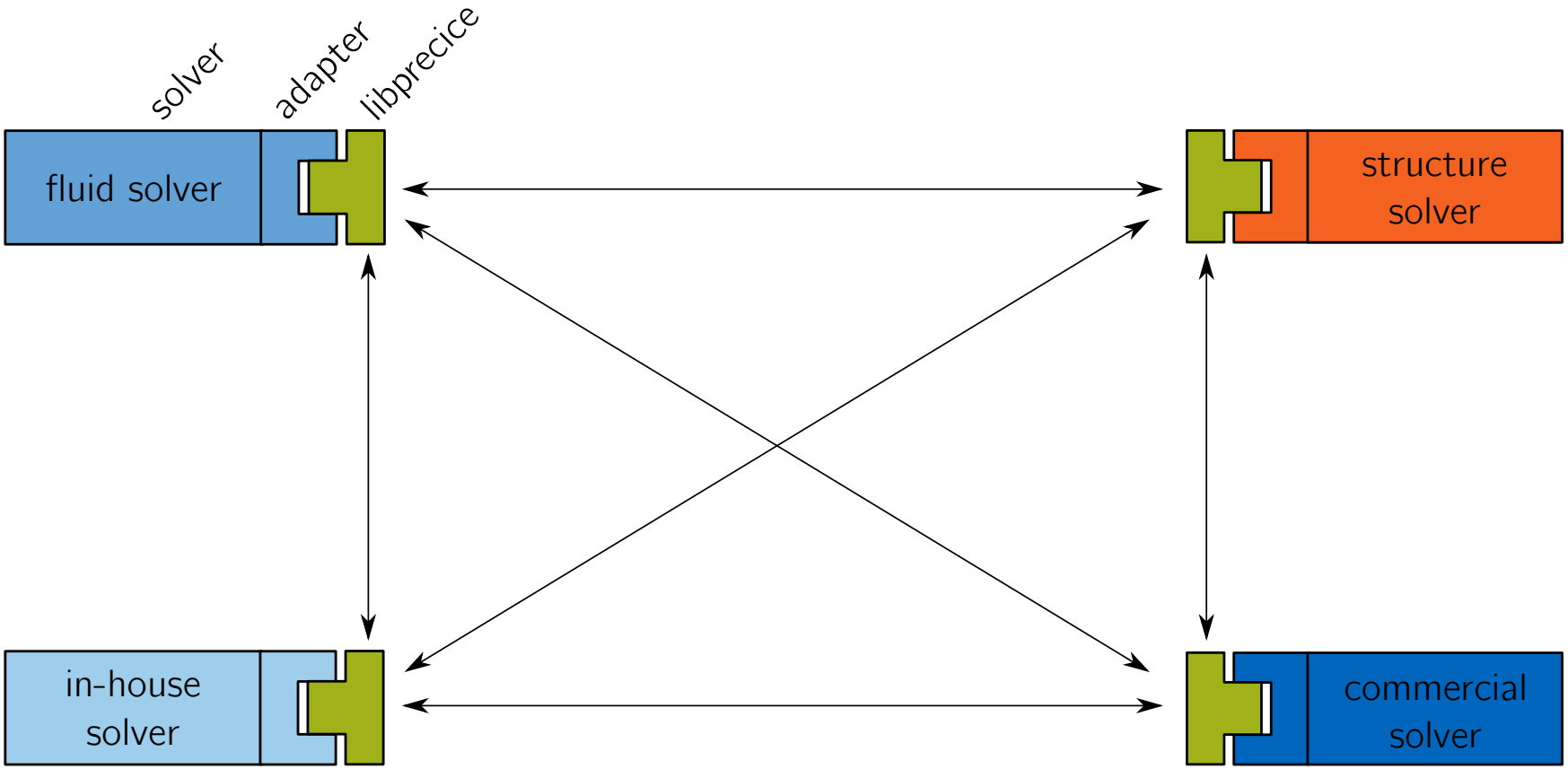
14th OpenFOAM Workshop - Duisburg, Germany
July 24, 2019



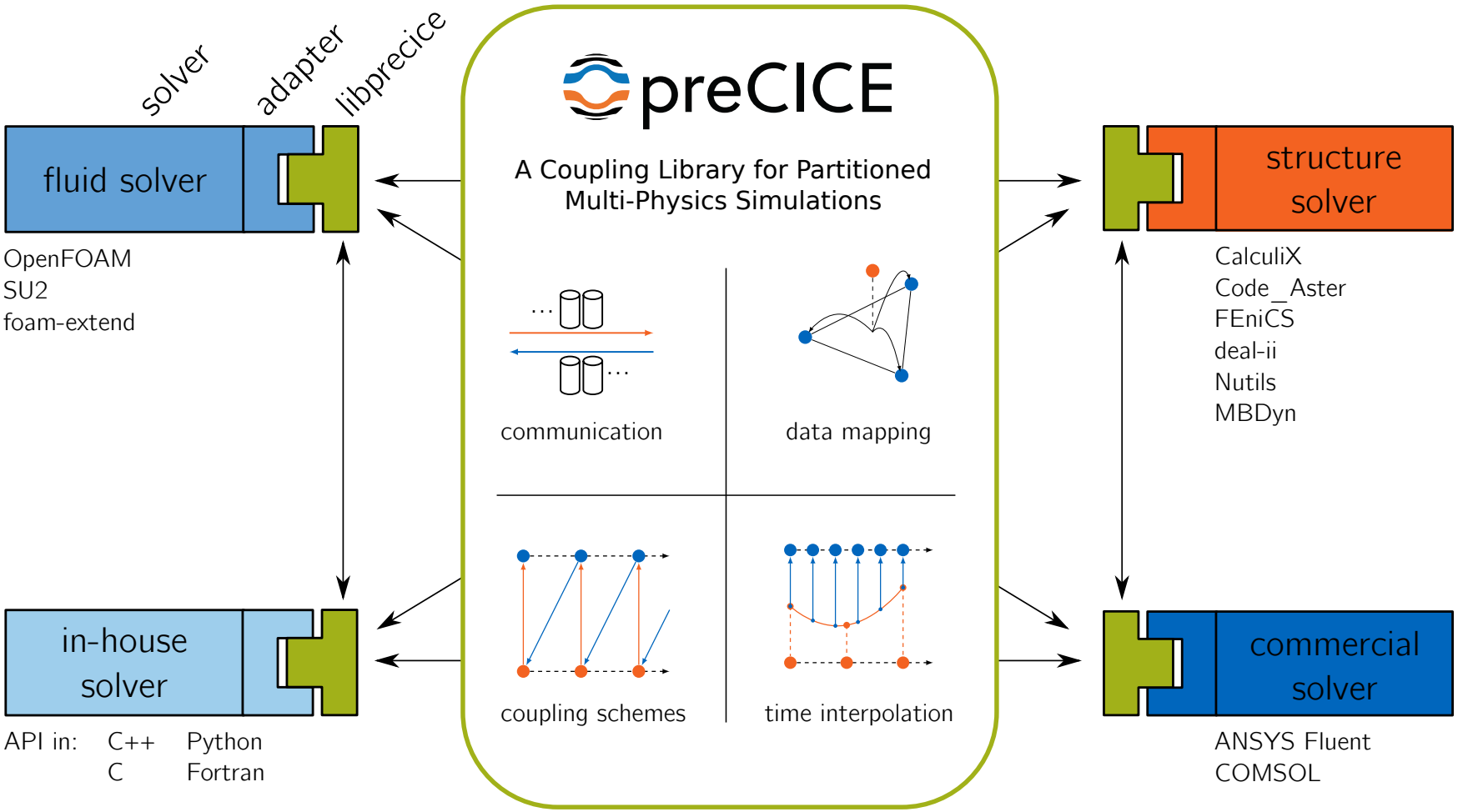
What is preCICE?

What is preCICE?

What is preCICE?



What is preCICE?



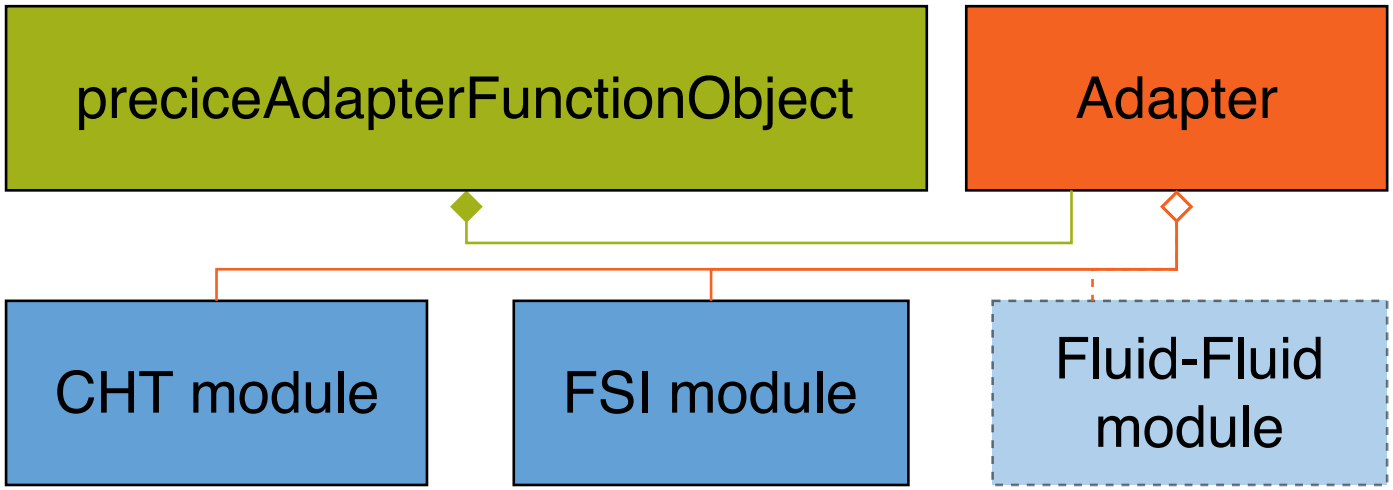
What is an adapter?

Adapted standard solver (example: CalculiX, SU2)

```
int main()
{
  /* Start the solver */

  while (runTime.run())
  {
      solve_equations();

      // Additional lines:
      precice.writeBlockVectorData( U );
      precice.advance();
      precice.readBlockScalarData( grad_p );
  }
}
```

OpenFOAM adapter: **plug-in** (function object)

One adapter, multiple physics

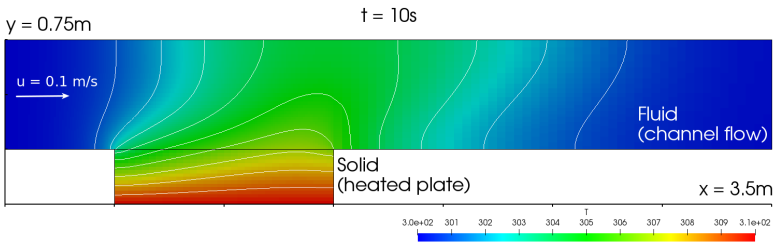


Conjugate heat transfer

Tutorials on `www.precice.org/resources` (step-by-step):

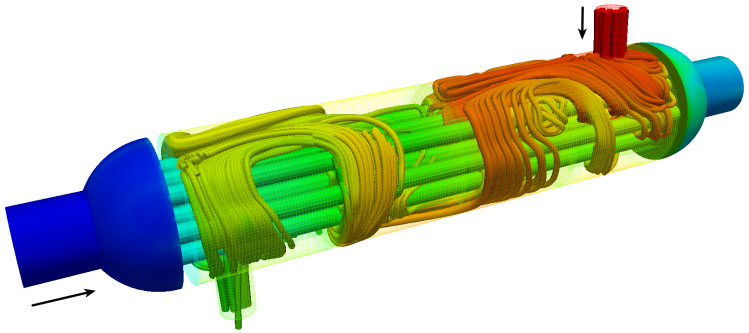
Flow above a heated plate

- Demo case, bundled with the adapter
- buoyantPimpleFoam + laplacianFoam
- Also: buoyantPimpleFoam + FEniCS
- Learn how to use the OpenFOAM adapter



Shell-and-Tube Heat Exchanger

- Larger case, in `precice/tutorials`
- buoyantSimpleFoam (x2) + CalculiX
- Learn how to do multi-coupling

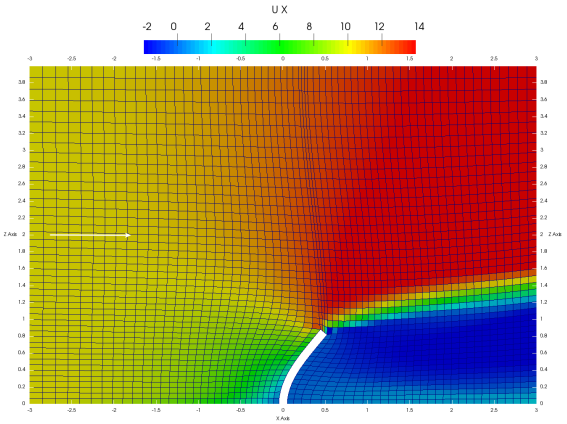


Fluid-structure interaction

Tutorials on `www.precice.org/resources` (step-by-step):

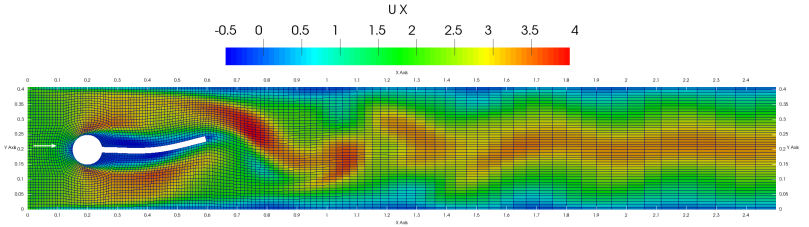
Flow in a channel with a perpendicular flap

- Quick simulation to get started
- Variant 1: pimpleFoam + CalculiX (non-linear)
- Variant 2: pimpleFoam + deal.ii (linear)



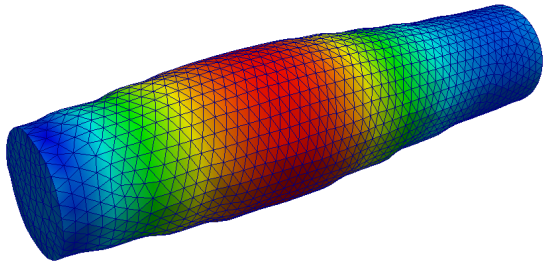
Turek & Hron cylinder + flap

- Larger case
- pimpleFoam + CalculiX (non-linear)
- Soon: pimpleFoam + FEniCS (non-linear)



Flexible 3D tube

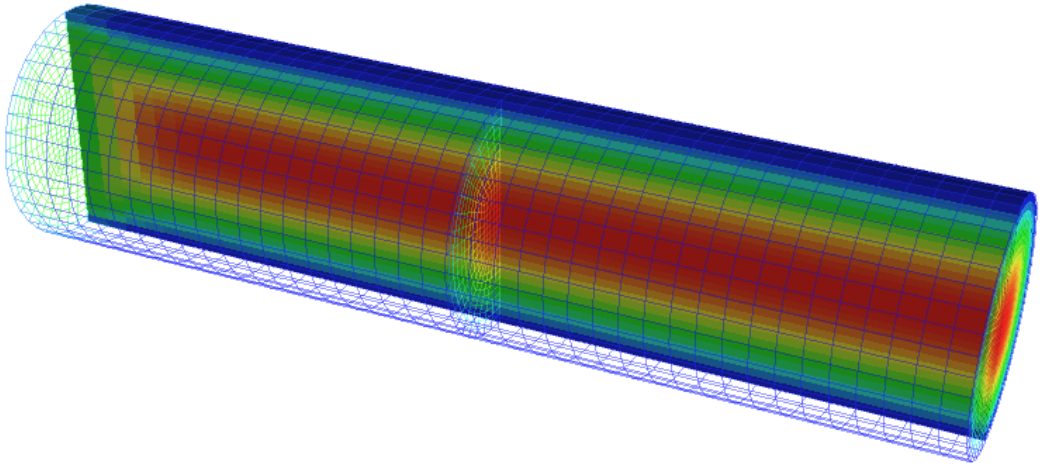
- Nearet-projection mapping
- pimpleFoam + CalculiX (non-linear)



Fluid-fluid coupling

- Experimental (branch `FF`)
- Currently supports:
  - Velocity
  - Pressure
  - Velocity gradient
  - Pressure gradient

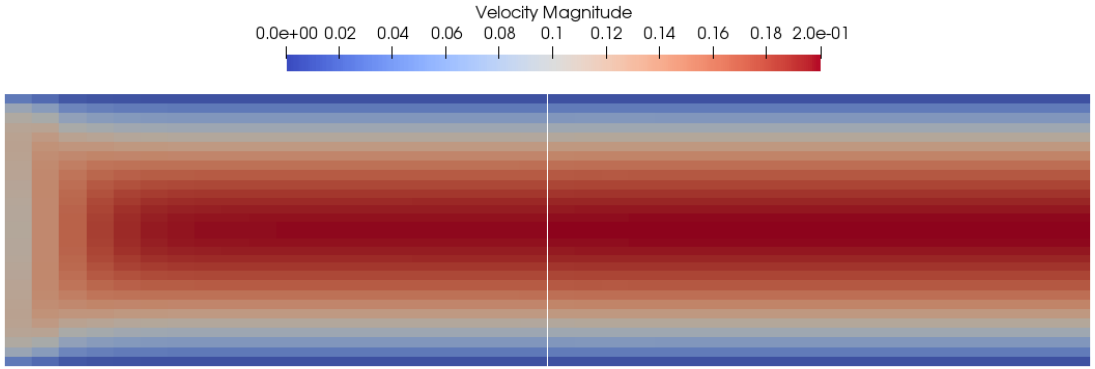
- Both incompressible and compressible
- No exchange of turbulence/multiphase fields



Fluid-fluid coupling: validation

- Pipe-pipe coupling (pimpleFoam)
- Implicit, bi-directional coupling

- z-axis symmetry, 40 m long, 10 m wide
- $\nu = 10\,\mathrm{m}^2/\mathrm{s}$, $\rho = 1\,\mathrm{kg}/\mathrm{m}^3$



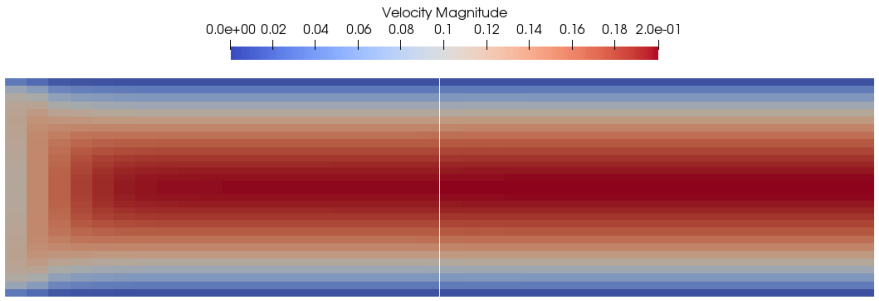
Left pipe:

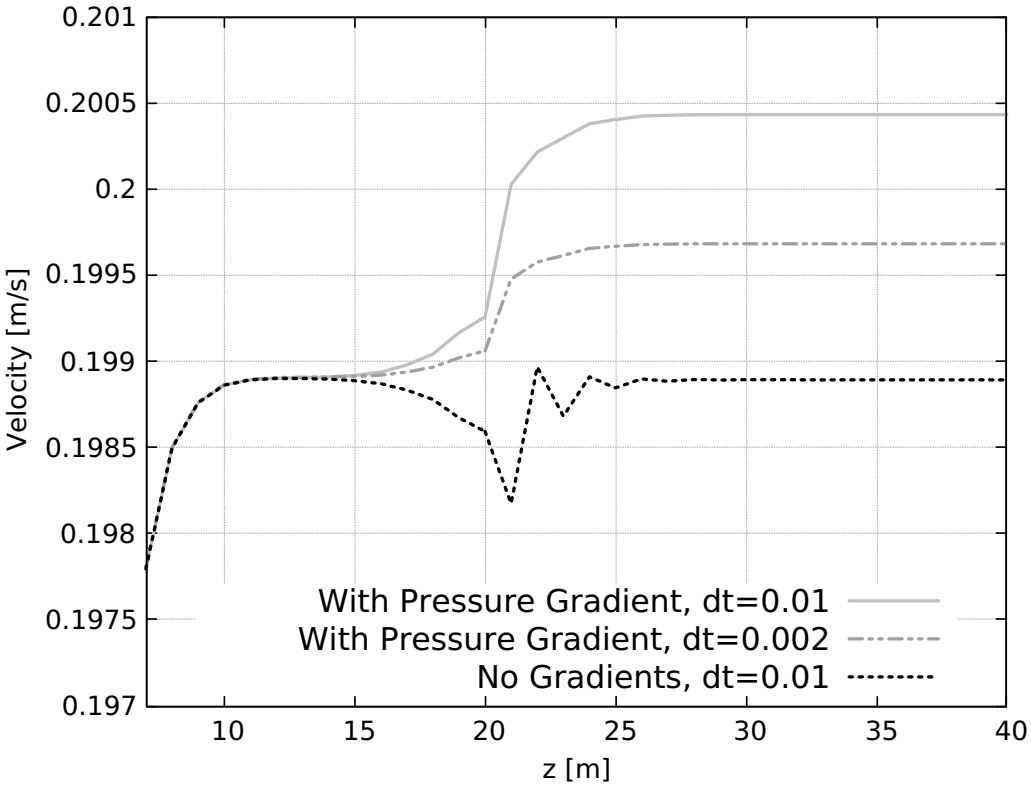
- Inlet: $U = 0.1\mathrm{m}/\mathrm{s}, \nabla p = 0$
- Outlet: $\nabla U = 0, p = 0$ (coupled)
- Writes: velocity [, pressure gradient]
- Reads: pressure

Right pipe:

- Inlet: $U = 0, \nabla p = 0$ (coupled)
- Outlet: $\nabla U = 0, p = 0$
- Writes: pressure
- Reads: velocity [, pressure gradient]

Fluid-fluid coupling: looking closer





Reading & writing pressure values

```
p->boundaryFieldRef()[patchID][i]
=
buffer[bufferIndex++];
```

```
buffer[bufferIndex++]
=
p->boundaryFieldRef()[patchID][i];
```

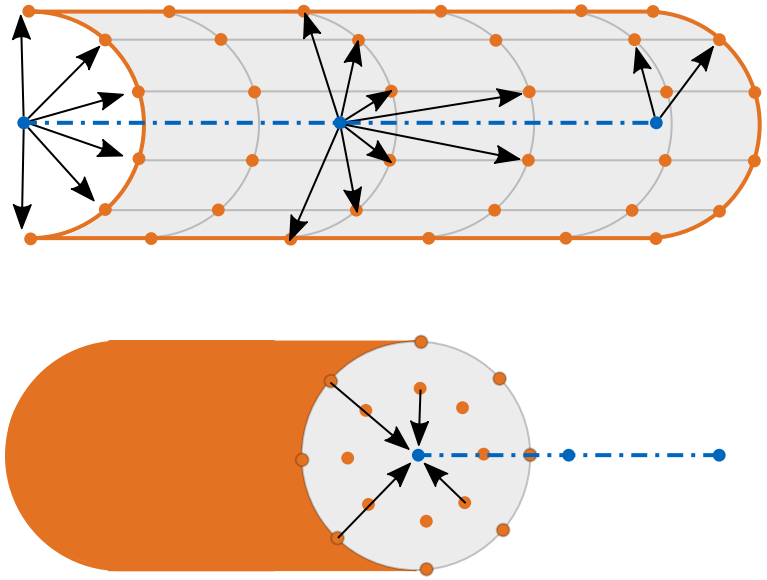
Reading & writing pressure gradients

```
scalarField & gradientPatch
=
refCast<fixedGradientFvPatchScalarField>
(
    p->boundaryFieldRef()[patchID]
).gradient();

forAll(gradientPatch, i)
{
    gradientPatch[i]
    =
    buffer[bufferIndex++];
}
```

```
scalarField gradientPatch
=
refCast<fixedValueFvPatchScalarField>
(
    p->boundaryFieldRef()[patchID]
).snGrad();

forAll(gradientPatch, i)
{
    buffer[bufferIndex++]
    =
    -gradientPatch[i];
}
```

Can you spot the bug? (if any here)

3D-1D mapping: terminology



$$\{\text{axial}, \text{radial}\} \times \{\text{collect}, \text{spread}\} \times \{\text{consistent}, \text{conservative}\} = 8 \text{ combinations}$$

axial-spread-consistent: $v_i^{3D} = v^{1D} \quad \forall i = 1 \dots n$

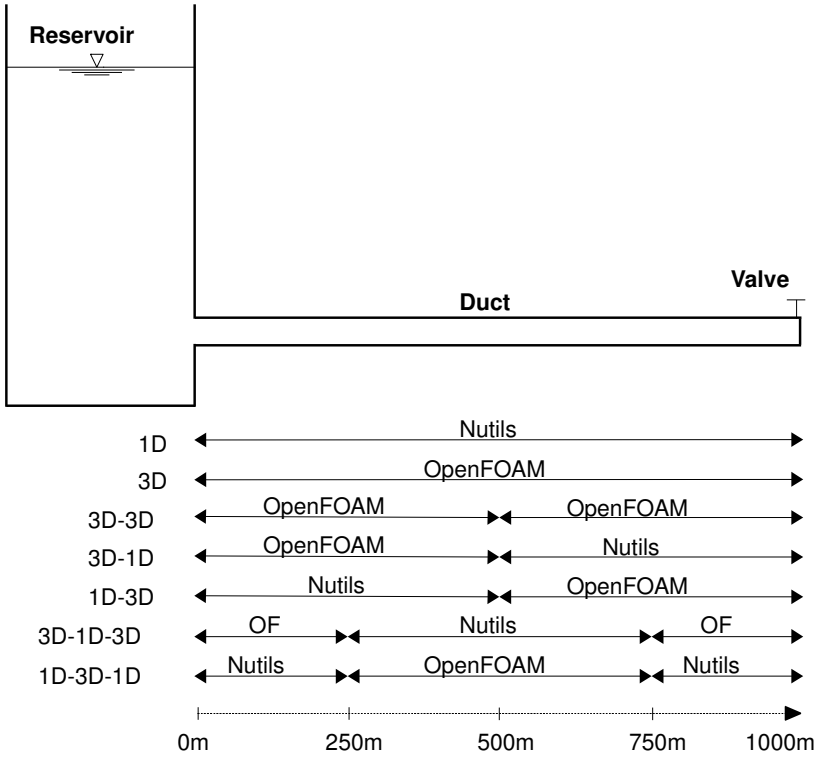
axial-collect-consistent: $v^{1D} = \frac{1}{n} \sum_{i=1}^{n} v_i^{3D}$

3D-1D: prototype in preCICE

```
<participant name="3D-Solver"> <!-- e.g. OpenFOAM -->
  <write-data name="Velocity" mesh="3D-Mesh"/>
  <read-data name="Pressure" mesh="3D-Mesh"/>

  <mapping:axial-geometric-multiscale
     type="spread" from="1D-Mesh" to="3D-Mesh" direction="read" constraint="consistent"/>
  <mapping:axial-geometric-multiscale
     type="collect" from="3D-Mesh" to="1D-Mesh" direction="write" constraint="consistent"/>
</participant>
```

Important: solvers themselves do not know whether they are coupled to a 1D or 3D partner!

3D-1D: validation setup

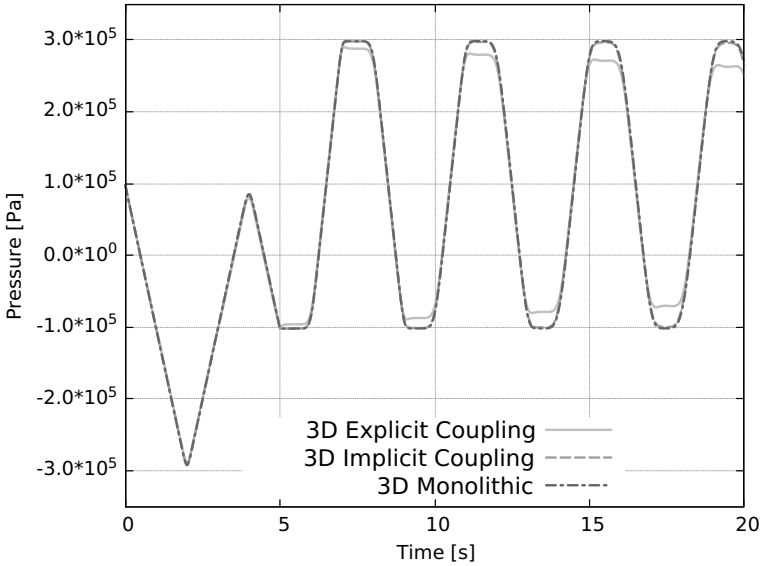


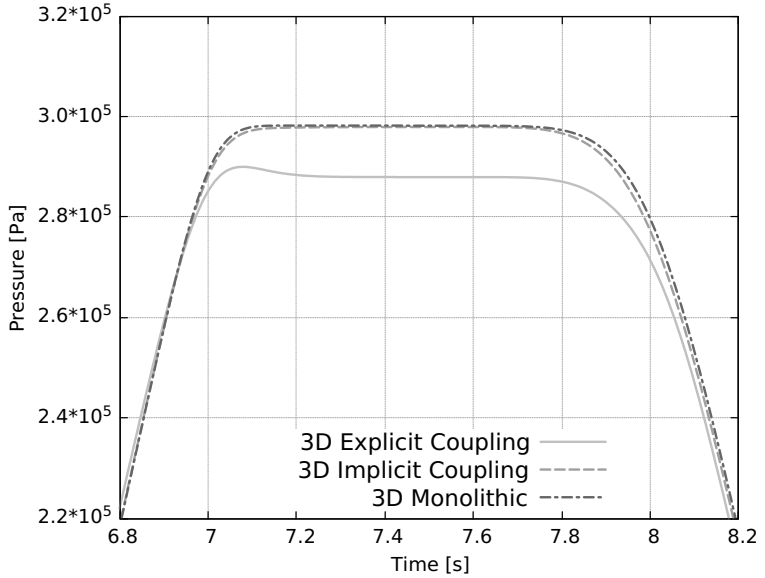
A reservoir-duct-valve system. A similar figure is already used by Wang et al.[1]

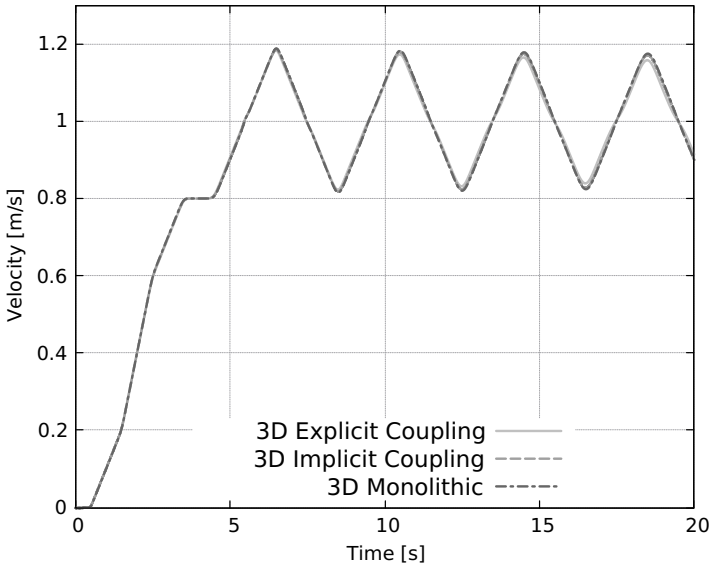
- barotropic, laminar, compressible fluid (sonicLiquidFoam)
- constant pressure at inlet of tube (infinite reservoir)
- increasing outlet velocity (opening valve)

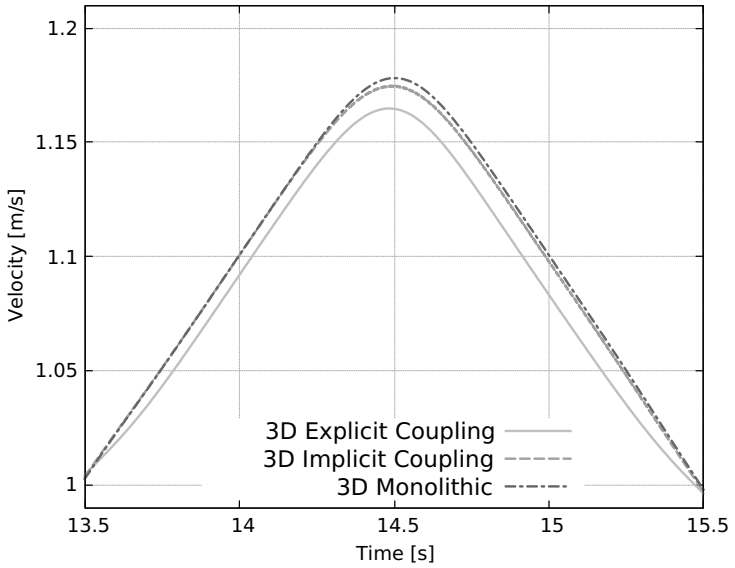
[1] C. Wang, H. Nilsson, J. Yang and O. Petit 1D-3D coupling for hydraulic system transient simulations. Computer Physics Communications, 210:1–9, 2017.

3D-3D: Explicit vs implicit coupling

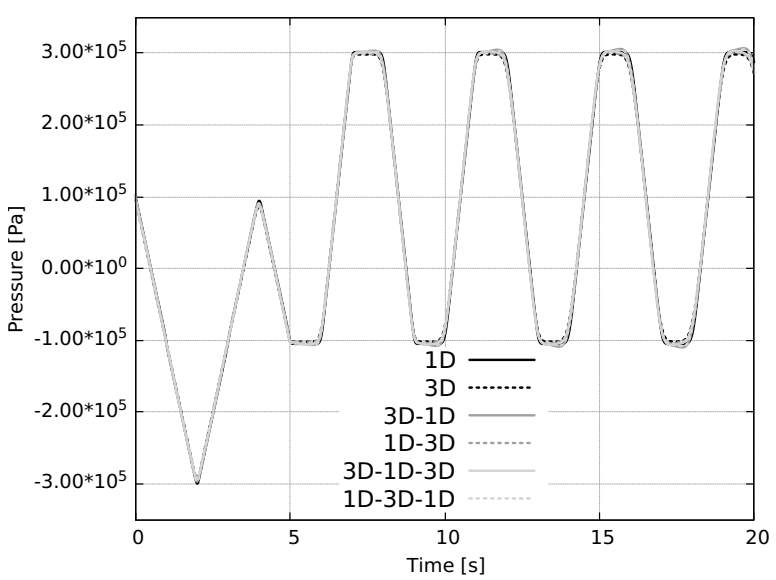


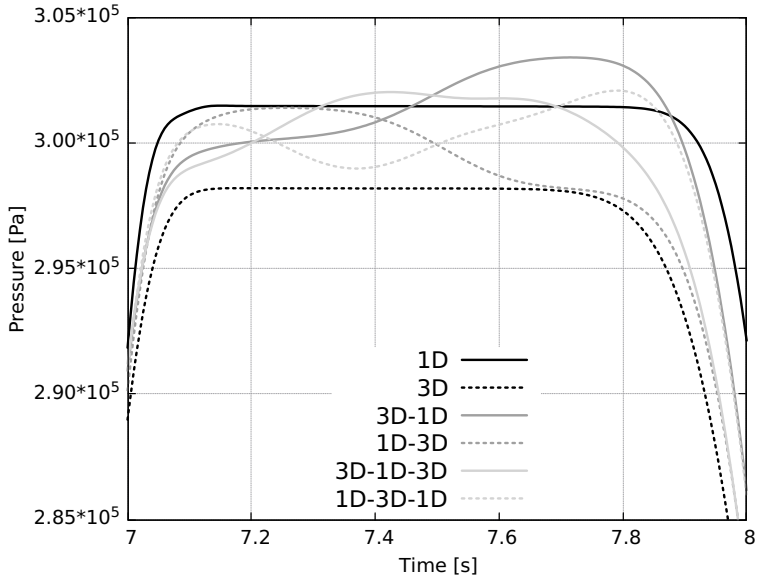






3D-1D-3D: comparison of setups





Further functionality

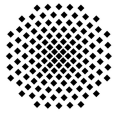
- Modifying the time step size
- Storing checkpoints (complete state & time)
- Restoring checkpoints (complete state & time)
- FSI: Moving the mesh

Acknowledgements

Thanks to Håkan Nilsson for the Water Hammer case setup.

preCICE is free because of the support of:

H2020 grant 754462



Universität
Stuttgart

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

Summary

preCICE + OpenFOAM / OpenFOAM The OpenFOAM Foundation

Already:	a plugin for CHT, FSI (incompressible)
Testing:	fluid-fluid, FSI (compressible)
Development:	3D-1D mapping
Compatible with:	OpenFOAM 4.0 - 7, v1706 - v1906
Help needed:	Supporting multiple versions (issue #32), Converting to an OpenFOAM dictionary (issue #30), Debugging FF (issue #93)

Summary

preCICE + OpenFOAM / OpenFOAM The OpenFOAM Foundation

Already:	a plugin for CHT, FSI (incompressible)
Testing:	fluid-fluid, FSI (compressible)
Development:	3D-1D mapping
Compatible with:	OpenFOAM 4.0 - 7, v1706 - v1906
Help needed:	Supporting multiple versions (issue #32), Converting to an OpenFOAM dictionary (issue #30), Debugging FF (issue #93)

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

Summary

preCICE + OpenFOAM / OpenFOAM The OpenFOAM Foundation

Already:	a plugin for CHT, FSI (incompressible)
Testing:	fluid-fluid, FSI (compressible)
Development:	3D-1D mapping
Compatible with:	OpenFOAM 4.0 - 7, v1706 - v1906
Help needed:	Supporting multiple versions (issue #32), Converting to an OpenFOAM dictionary (issue #30), Debugging FF (issue #93)

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).

Backup: fvSchemes

```
gradSchemes
{
    default         Gauss linear;
}

snGradSchemes
{
    default         corrected;
}
```