NATIONALES HOCHLEISTUNGS RECHNEN

KISSKI
KI-Servicezentrum für sensible und kritische Infrastrukturen

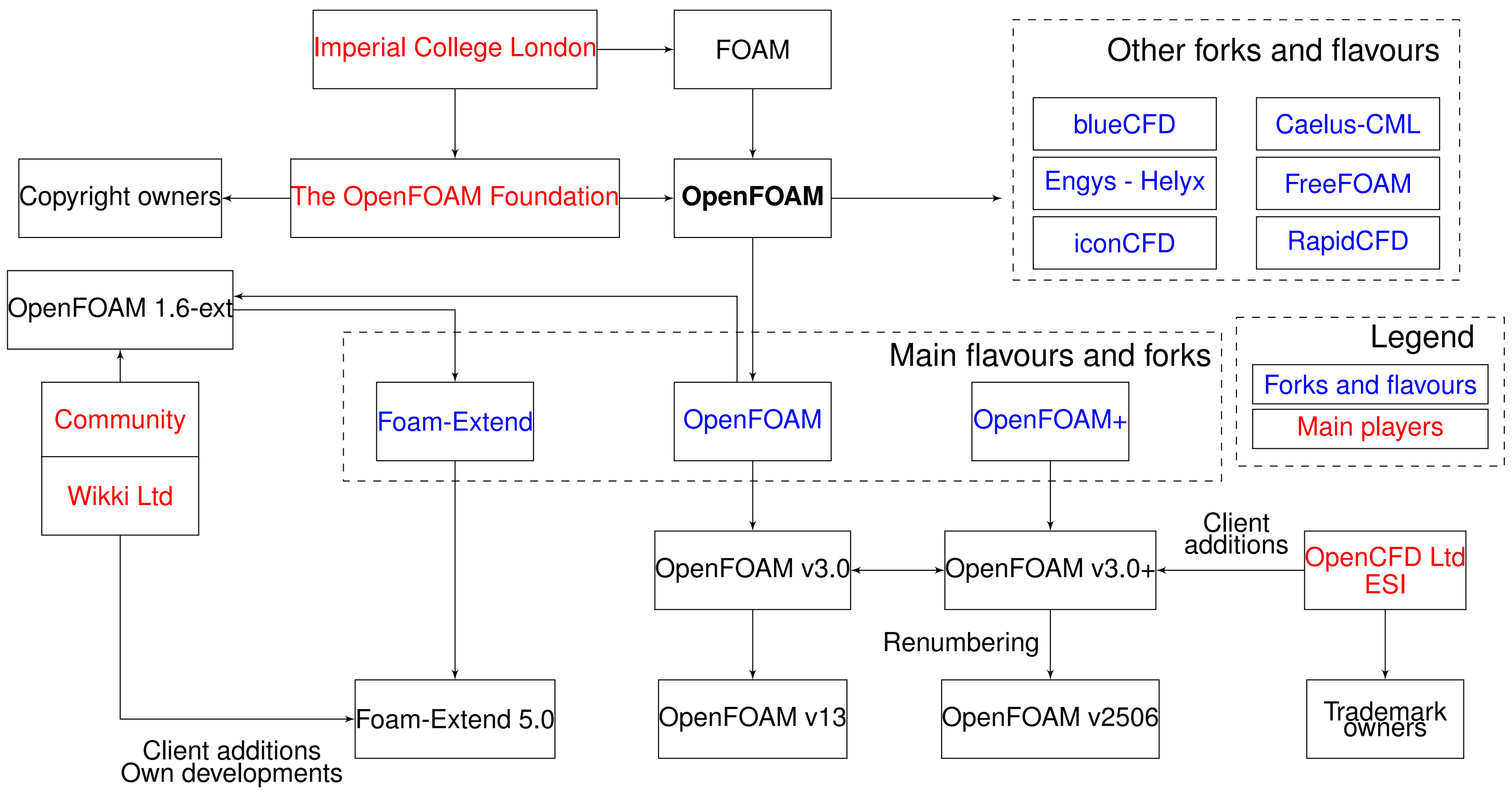
Development of a preCICE adapter for foam-extend

preCICE at NHR-NORD@Göttigen

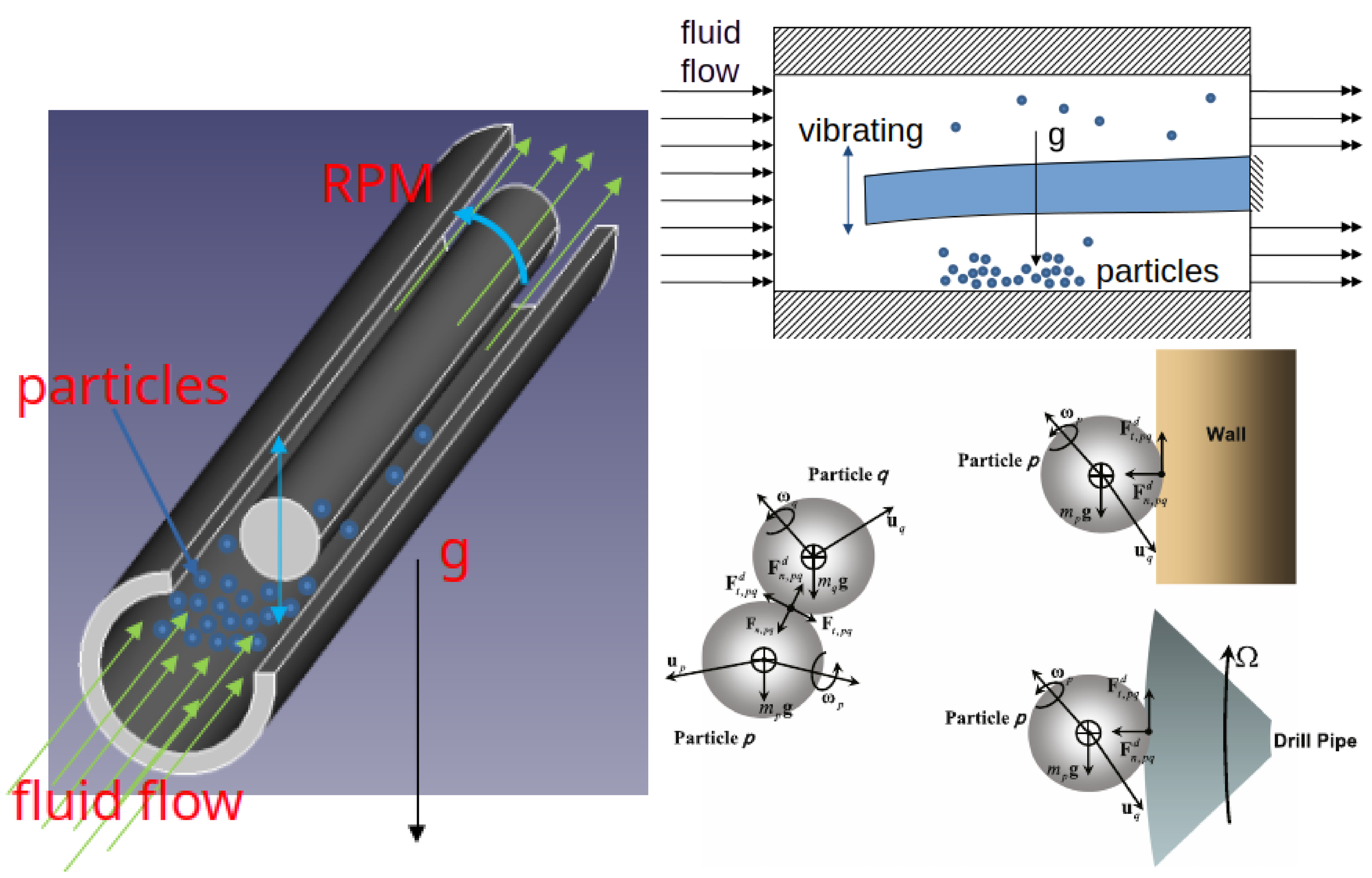
- Installed modules in cluster
- Development of foam-extend-adapter
- Currently packaging CalculiX, CalculiX-adapter and foam-extend-adapter in spack
- Ultilized in user-trainings

Background

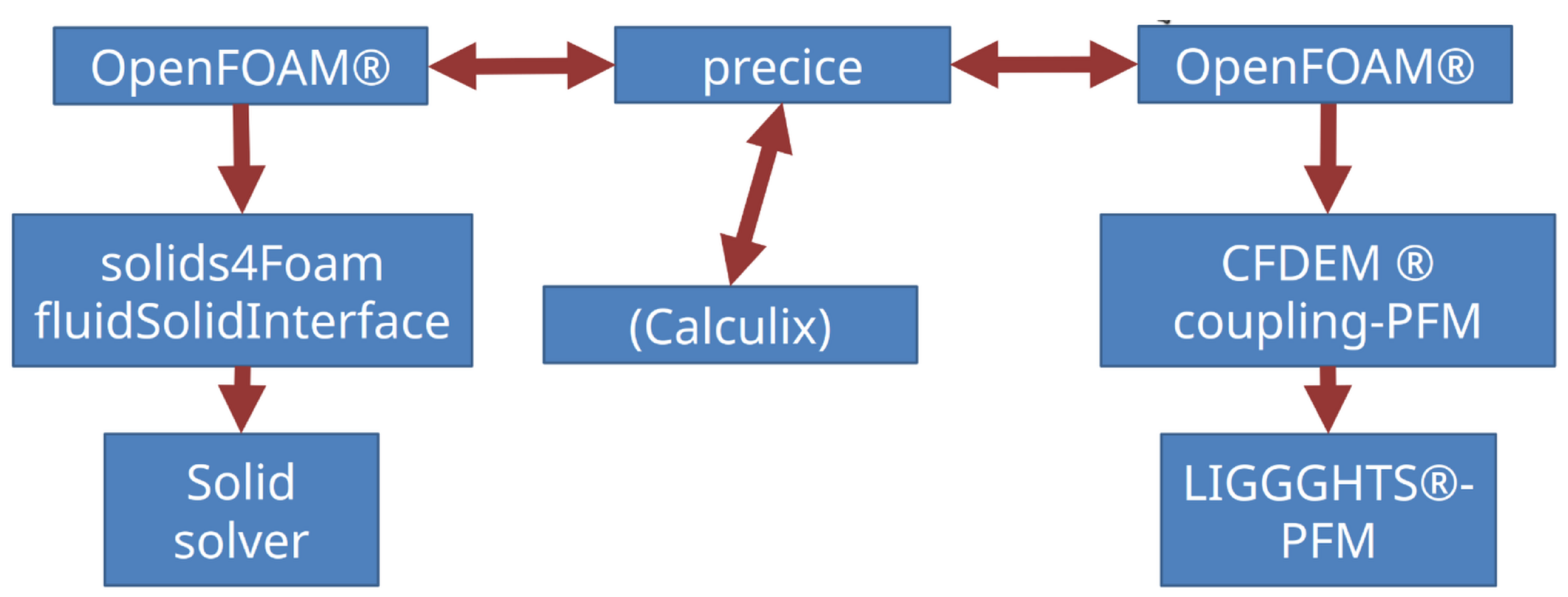
OpenFOAM Universe



Simulation Problem



Software Setup



Key Takeaways

- Prototype implementation working for 'quick-start' tutorial and foam-extend-5.0 and foam-extend-dev
- Open issues about implementation details, e.g. missing fields in ObjectRegistry
- Testing using inside Docker containers for reproducibility
- Experimental code available at `https://github.com/precice/foam-extend-adapter`
- Contact: Patrick Höhn (patrick.hoehn@gwdg.de)

Who are we?

GEORG-AUGUST-UNIVERSITÄT GÖTTINGEN IN PUBLICA COMMODA SEIT 1737

GWDG

- Co-owned by Max Planck Society and University of Göttingen
- National Center for High Performance Computing

NHR-NORD@GÖTTINGEN

- National Center for AI Services
- Operator of AcademicCloud and ChatAI

Development Approach

- Based on existing OpenFOAM-Adapter
- Differences between OpenFOAM flavors and within versions
- Inclusion of different versions in one source:
  - Via ifdef preprocessor includes
  - **Code separated in different git branches**

Challenges / Code differences

Compile Time errors

- API incompatibilities, e.g.:

```
-                   auto writeData = interfaceDict.get<wordList>("writeData");
+                   auto writeData = wordList(interfaceDict.lookup("writeData"));
```

```
-         const_cast<volScalarField::Internal&>(
+         const_cast<volScalarField::DimensionedInternalField&>(
```

```
-        addCheckpointField(mesh_.thisDb().getObjectPtr<GeomField>(obj)); \
+        addCheckpointField(mesh_.thisDb().lookupObject<GeomField>(obj)); \
```

- Changes in Header files, e.g.:

```
- #include "Time.H"
+ #include "foamTime.H"
+ #include "volFieldsFwd.H"
  #include "fvMesh.H"
+ #include "fileName.H"
```

- Naming collisions (same name of Header file for incompressible and compressible flows), e.g. split of CHT/HeatFlux in base, compressible and incompressible
- Changes of library names:

```
+     -lfoam \
      -lfiniteVolume \
      -lmeshTools \
-     -lcompressibleTurbulenceModels \
-     -lincompressibleTurbulenceModels \
-     -limmiscibleIncompressibleTwoPhaseMixture \
+     -lcompressibleTurbulenceModel \
+     -lincompressibleTurbulenceModel \
+     -lincompressibleTransportModels \
+     -lbasicThermophysicalModels \
```

- Missing components, e.g. ClockValue -> porting from OpenFOAM

Run Time errors

```
+           static IOdictionary preciceDict(
+               IOobject(
+                   "preciceDict",
+                   mesh_.time().system(),
+                   mesh_,
+                   IOobject::MUST_READ_IF_MODIFIED,
+                   IOobject::NO_WRITE));
            const dictionary& FSIDict =
-               mesh_.db().lookupObject<IOdictionary>("preciceDict").subOrEmptyDict("FSI");
+               preciceDict.subOrEmptyDict("FSI");
```