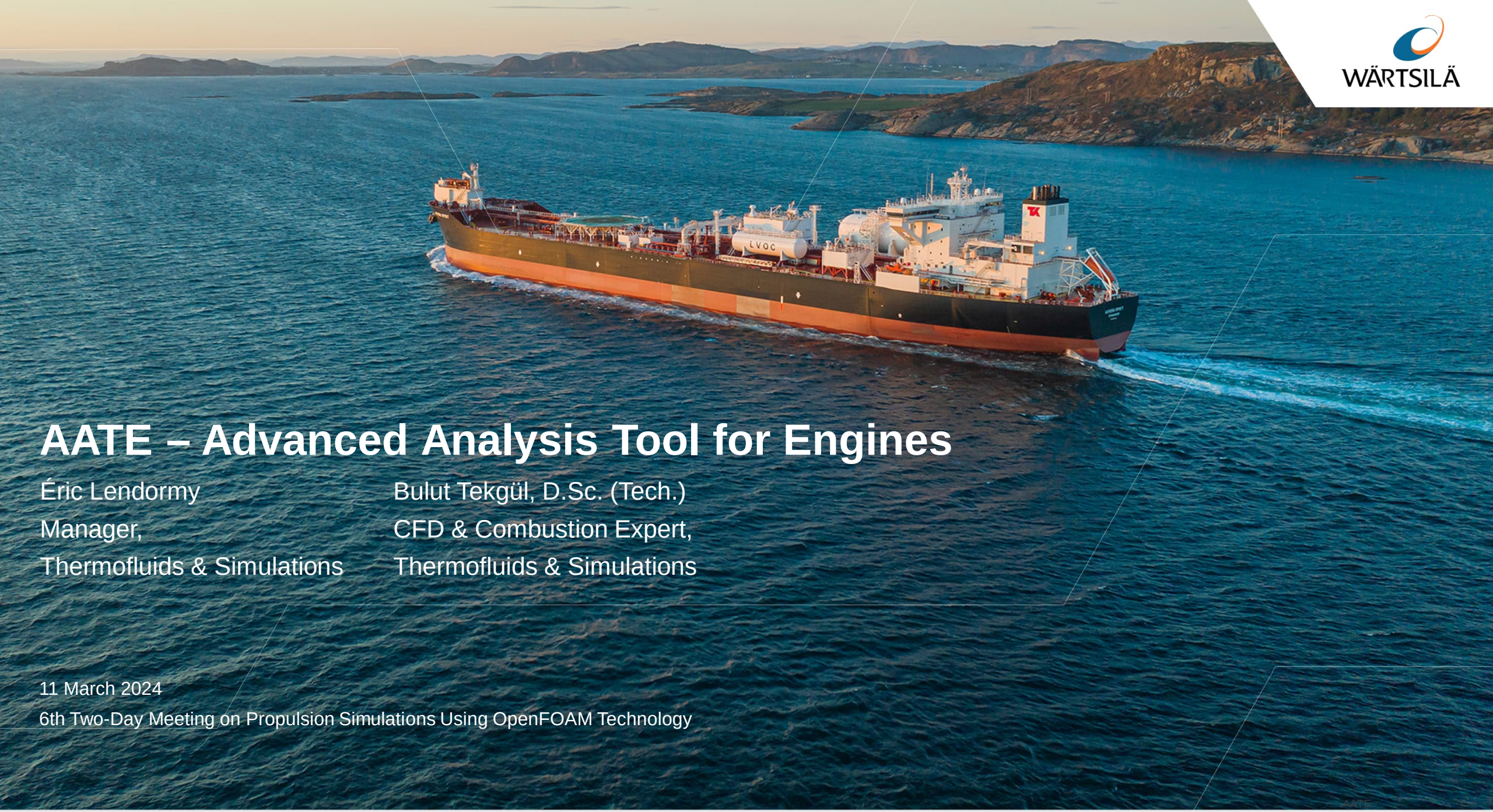


AATE – Advanced Analysis Tool for Engines

Éric Lendormy
Manager,
Thermofluids & Simulations

Bulut Tekgül, D.Sc. (Tech.)
CFD & Combustion Expert,
Thermofluids & Simulations

11 March 2024

6th Two-Day Meeting on Propulsion Simulations Using OpenFOAM Technology

Global leader

in decarbonising marine and energy

Founded in

1834

Net sales, MEUR

5,842

Our personnel

17,581

Nationalities

127

Country presence

79

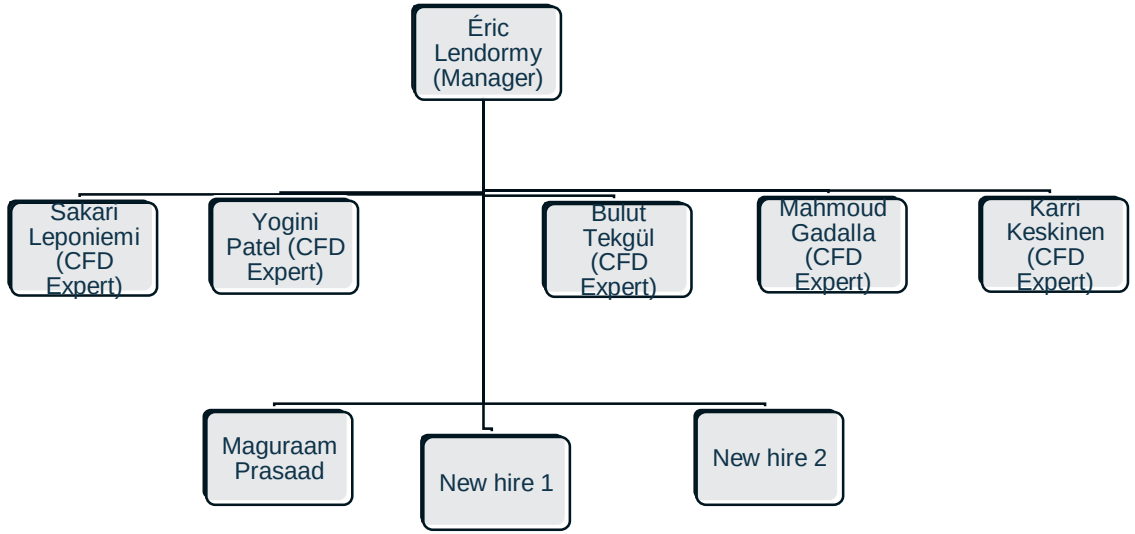
Locations worldwide

242

Figures from 2022

Thermofluids & Simulations

Who are we?



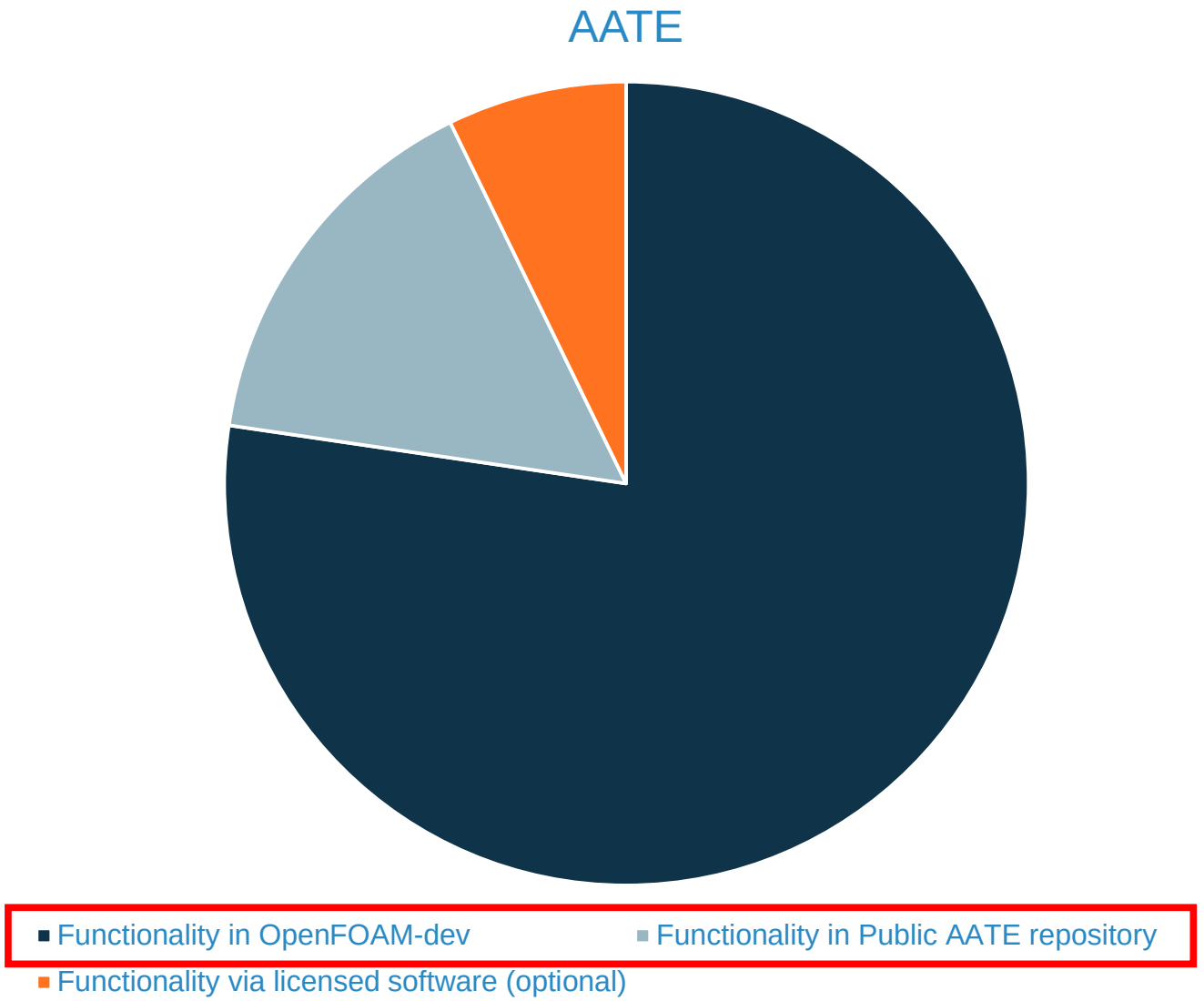
- **Cold flows** : Liquid (Oil, water, fuel) and gases (gas fuels, air)
  - Solves cavitation, THC emissions or pressure force and loss problems
- **Reacting flows** : Combustion (Diesel, SG, DF, HCCI, future fuels); catalysts (SCR)
  - Improves combustion performance while keeping emissions low
- **Heat flows** : Heat transfer and conductivity
  - Improves durability and prediction of components' lifetime

What is AATE?

AATE (/ˈɑːteˣ/): Advanced Analysis Tool for Engines

- OpenFOAM-based framework for engine simulations
- Product of 3-year collaboration with CFD-Direct
- New, efficient, and robust functionalities in OpenFOAM
- **Free and open-source, and part of OpenFOAM**

Content of AATE



A brief recap of Wärtsilä – CFD Direct collaboration

1. Test case preparation.
2. Generalise the engineFoam solvers by e.g. merging them into the rhoPimpleFoam type solvers.
3. Re-design the fvMotionSolverEngineMesh class to enable mesh motion for valves and piston.
4. Provide an essential functionality to execute mesh-to-mesh mapping whenever necessary.
5. Assess the potential future development routes to enhance the workflow.

Friday 15th October, 2021
Heikki Kahila

- Generalisation of engineTime userTime option (Oct 19 2021)
- Mesh to mesh mapping (April 6 2022)
- Non-conformal coupling (funded primarily by VTT, Finland) (May 9 2022)
- Engine mesh mover and test cases (12 February 2024)

Year	Maintenance (EUR)	Development (EUR)	Total (EUR)
2024	25,000	50,000	75,000
2023	25,000	60,000	85,000
2022	20,000	40,000	60,000
2021	20,000	20,000	40,000

Cost does not scale with number of users

Incentive to maximize the usage

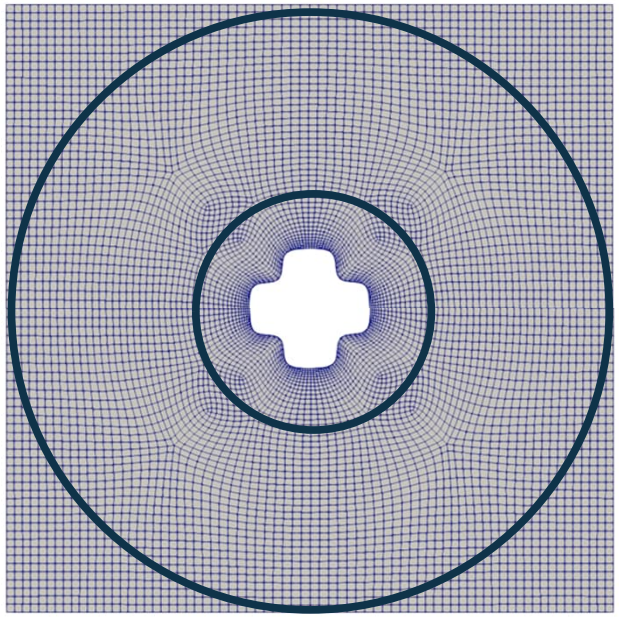
Core Development (so far)

Challenge	Solution	Publicly available?
Mesh needs to deform with combustion chamber and valve geometries	▪ fvMeshMovers::multiValveEngine **mesh mover** ▪ Internal development, available in OpenFOAM-dev	YES ✓
Complex geometries benefit from modularity	▪ **Non-conformal coupling (NCC)** interfaces ▪ Mesh modules can be constructed independently and with arbitrary tools ▪ Interfaces slide over one another – preserve mesh quality ▪ Funded by Process Engineering Consortium	YES ✓
Single mesh cannot handle geometries with different dimensions	▪ **Mesh-to-mesh mapping** ▪ Seamless on-the-fly transition between computational grids ▪ Developed by OpenFOAM developers through Wärtsilä budget	YES ✓
Anything else we need	▪ **Meshing strategies, Injection models, novel quantities, postprocessing tools…**	PARTIALLY

Mesh mover - how does it work?

- Idea based on `interpolatingSolidBodyMotionSolver.C`

1. Find the translation / rotation matrix induced by the moving object.
2. Define ”inner” and ”outer” distance to restrict the interpolated motion.
3. Define a scaling function [0,1], normalised between inner and outer distances.
4. Move the mesh points ***explicitly*** according to the scale function in their respective position. No implicit equation solved.

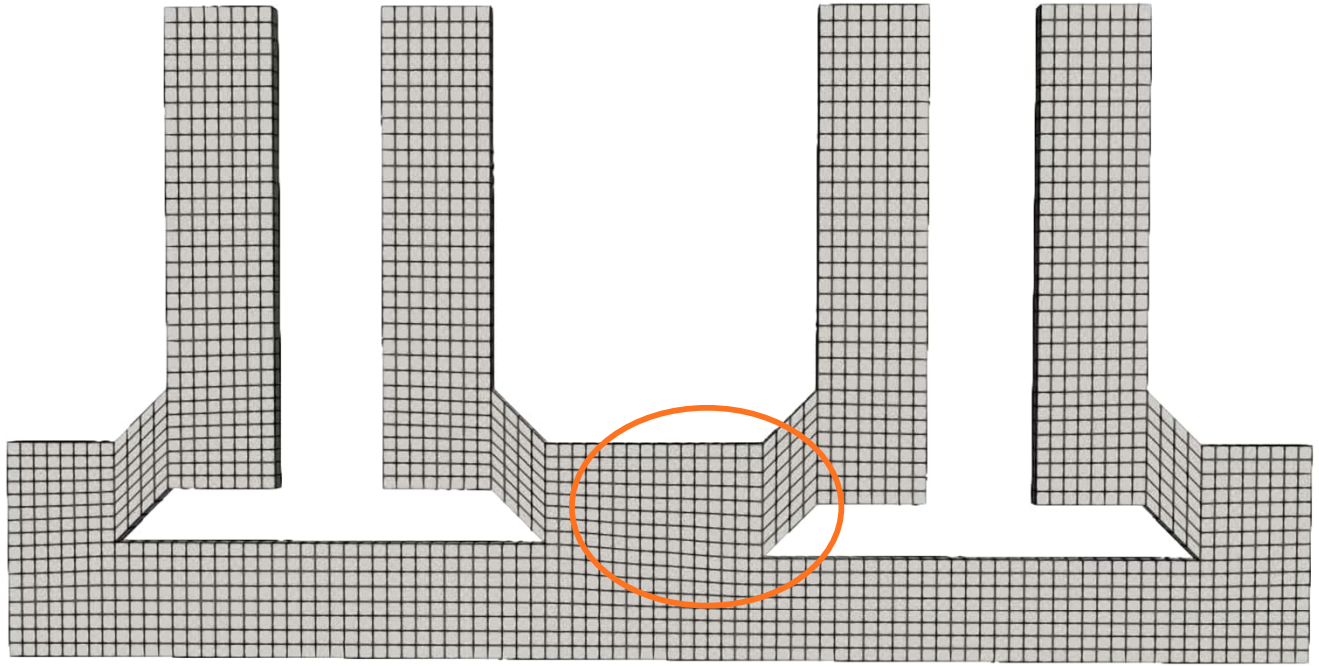


Mesh mover – I/O

```
piston
{
    patches             (piston);
    axis                (0 0 1);
    movingZones         (pistonBowl);
    frozenZones         (cylinderHead prechamber);
    motion
    {
        type            crankConnectingRodMotion;

        // Mimic pure sinusoidal motion
        conRodLength    1000;
        stroke          1;
    }

    movingFrozenLayerThickness  0;

    fractionalTravelInterval    0.1;
    cosineScaling               yes;
}
```

```
valves
    {
        iv
        {
            patches         (ivHead);

            axis            (0 0 1);

            movingFrozenLayerThickness  0;

            fractionalTravelInterval    0.02;
            cosineScaling               yes;

            minLift                     0.01;

            motion
            {
                type    table;
                values
                (
                    // lift: 0.002 / CAD
                    (0      0.0)
                    (340    0.01) // IVO + meshToMesh map
                    (480    0.29)
                    (620    0.01) // IVC + meshToMesh map
                    (720    0.0)
                );

                // For multi-cycle simulations, use repeat
                outOfBounds     repeat;
                interpolationScheme linear;
            }
        }
     }
```

Mesh mover

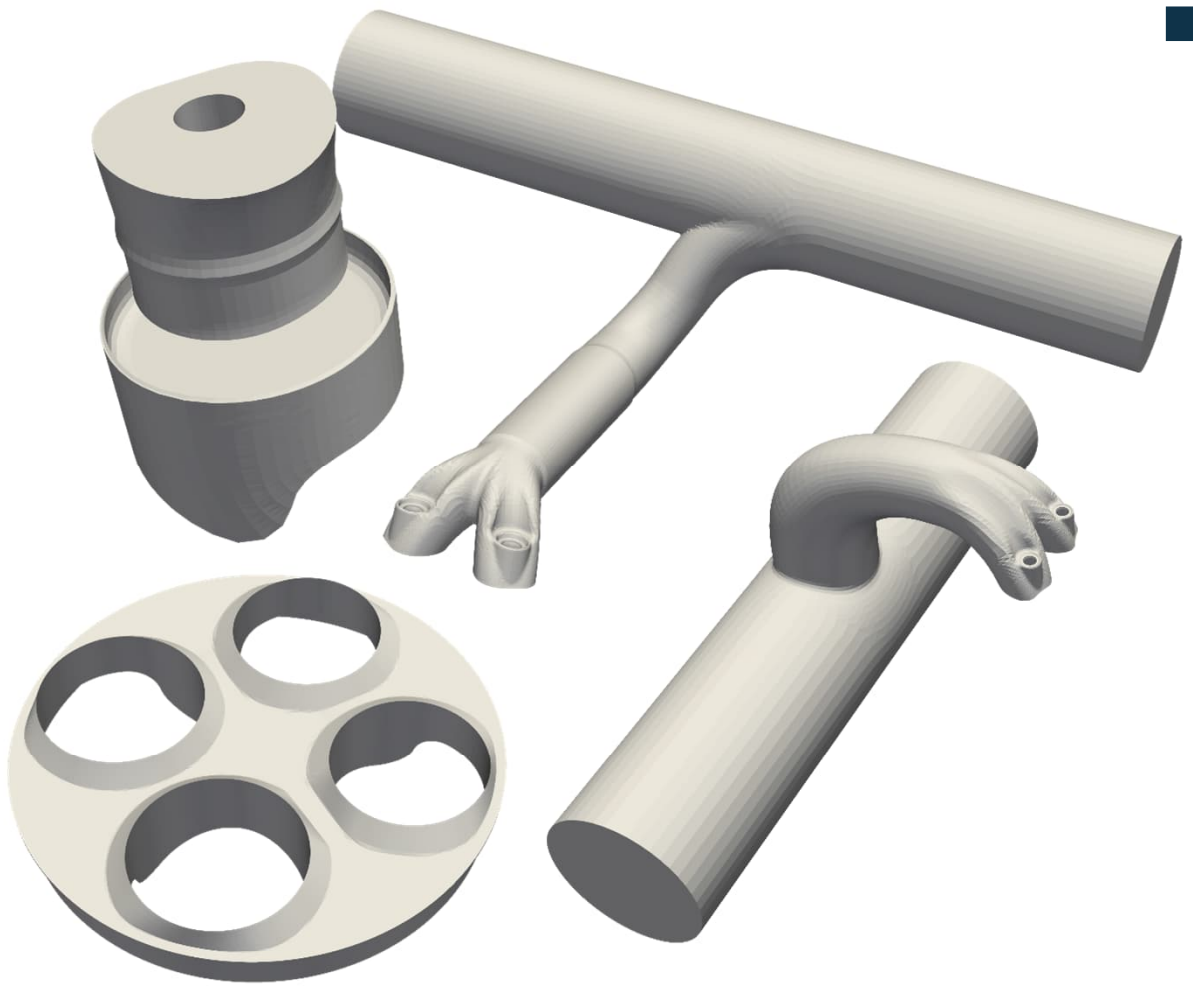


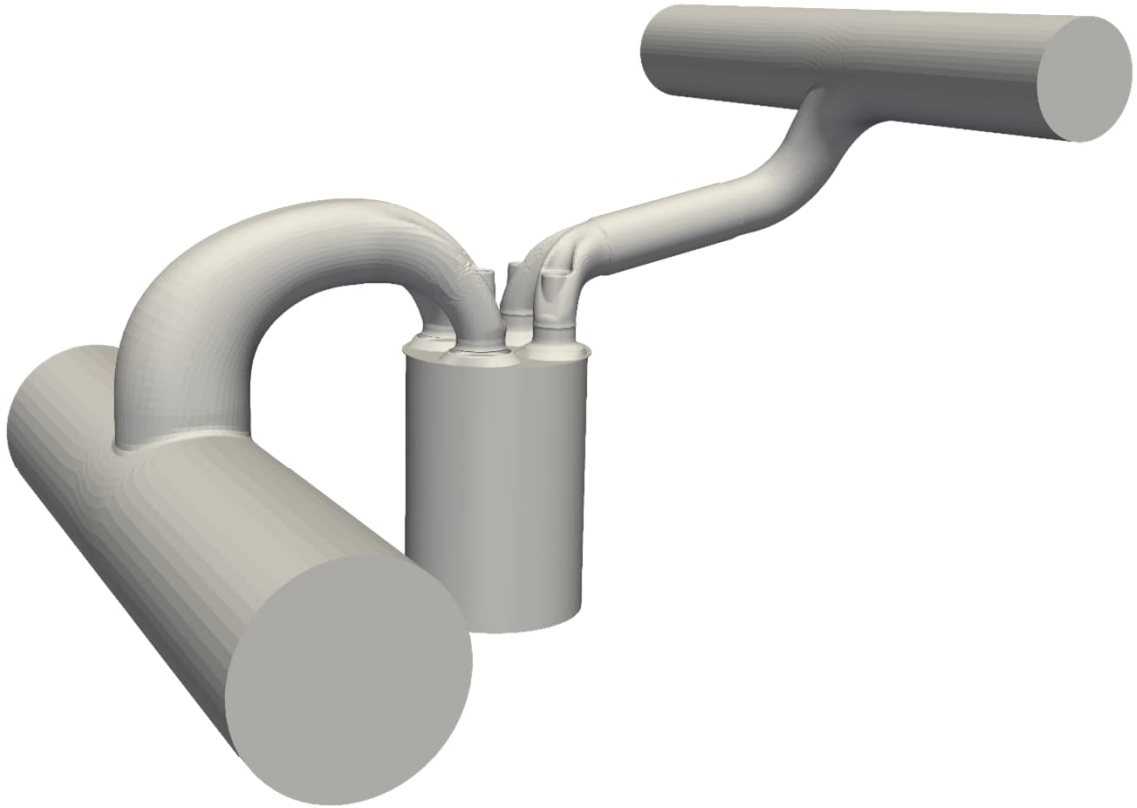
Modular meshing approach via Non-Conformal Coupling (NCC)

Mesh modules

Merged meshes
+
NCC

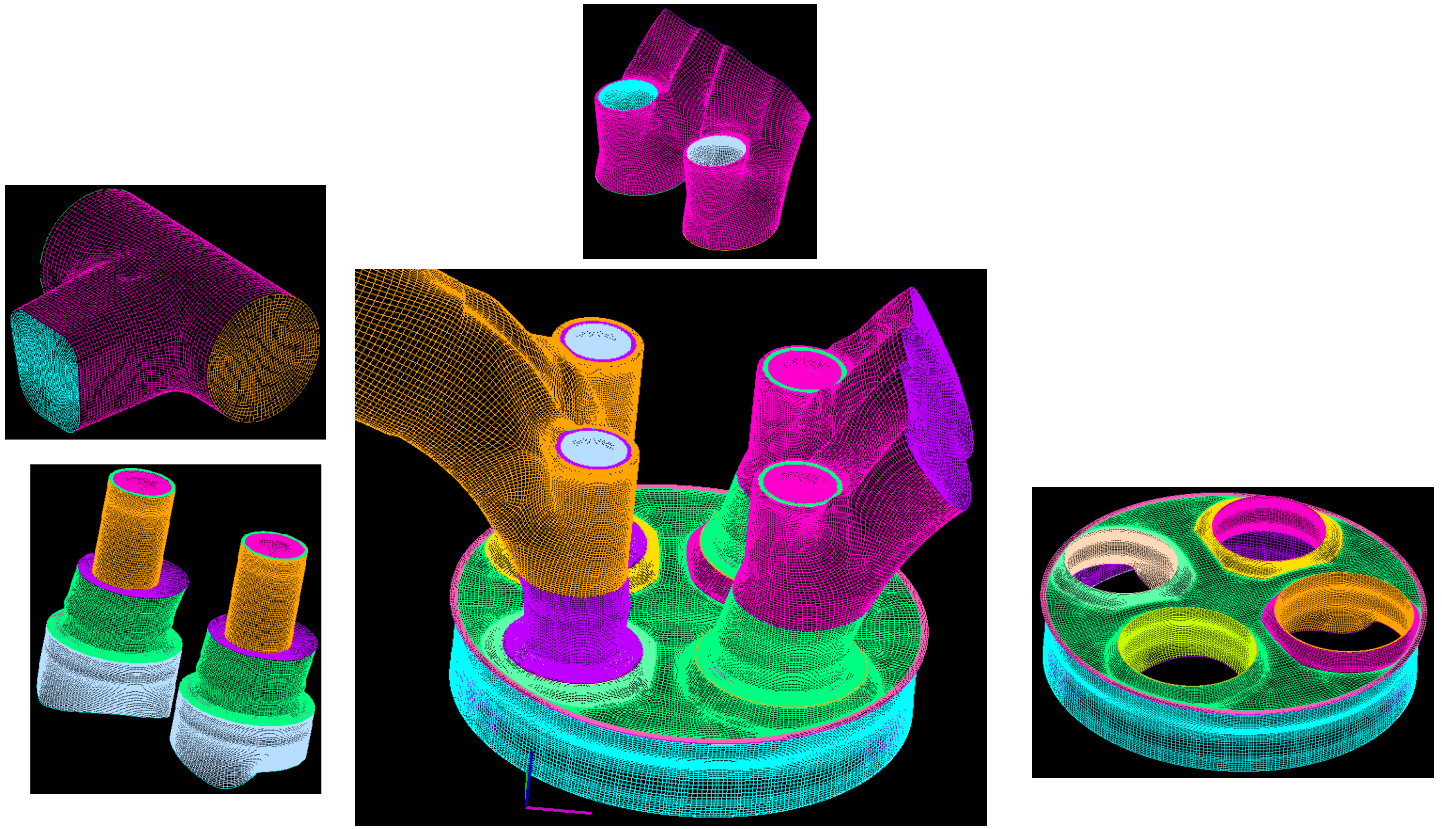
Final mesh





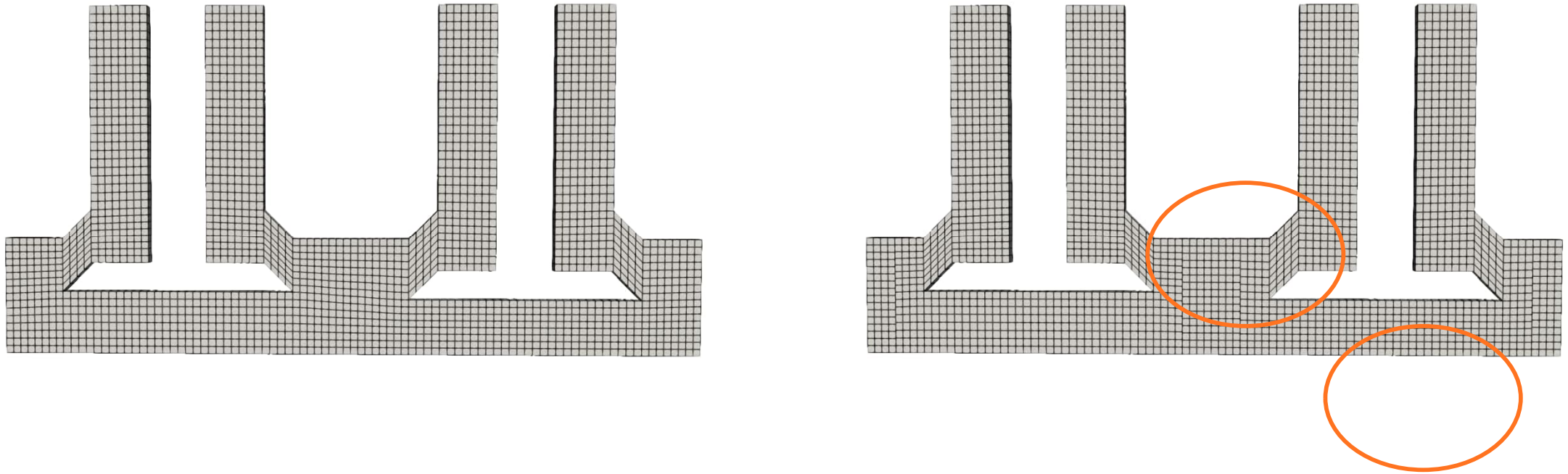
Modular meshing approach via NCC

- Fully structured 3D meshes can be generated
- Robust, accurate, and fast solutions
- User coding required to establish proper meshing automation: possible via python/shell API



Mesh mover

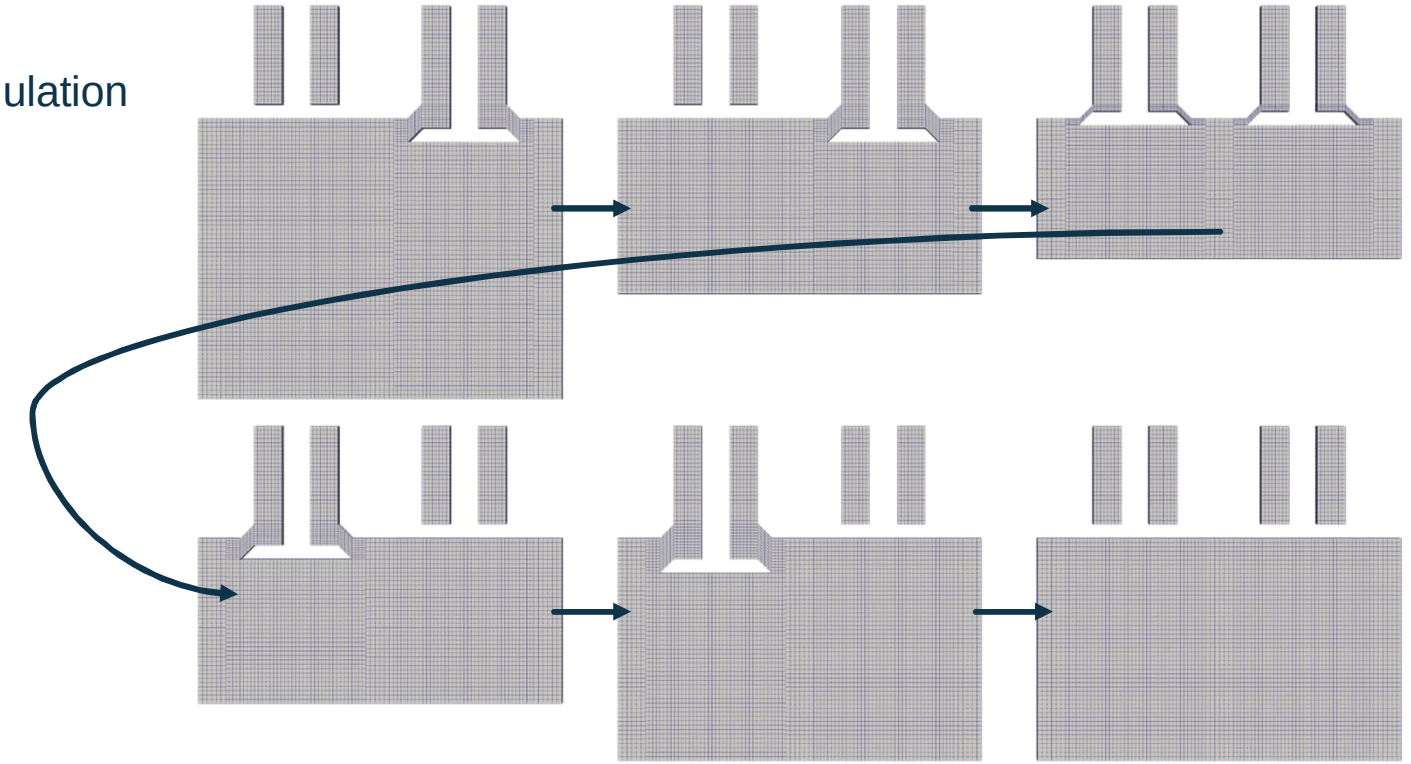
Standard mesh deformation → Mesh deformation with NCC boundaries



Mesh mapping

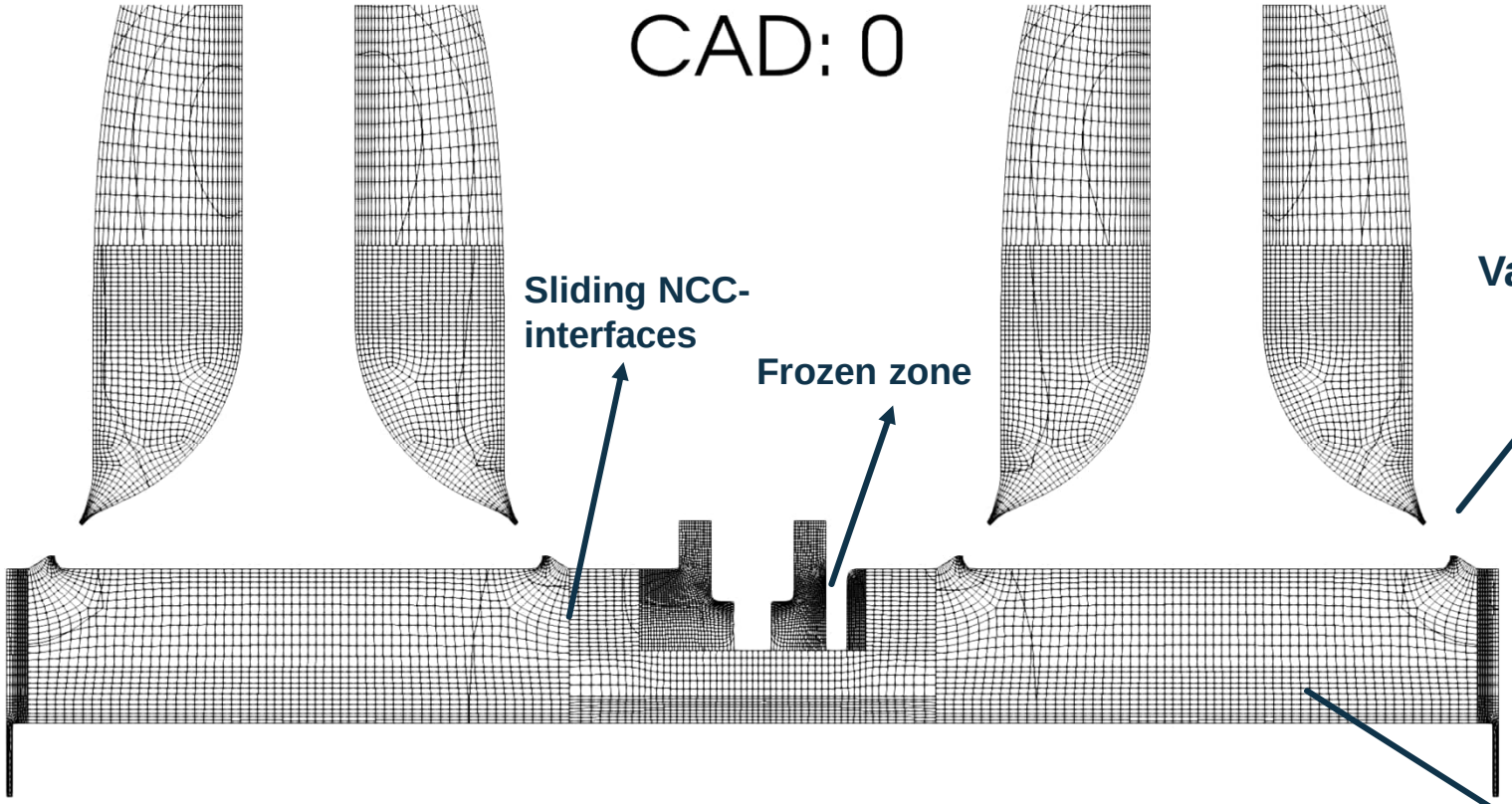
What do we do when mesh moves ”too much”?

- Too much deformation leads to low quality mesh
- Mesh is switched to a ”fresh” instance during simulation
- Mesh instances are **pre-generated**



Real engine application

Mesh to mesh mapping + smooth valve and piston motion



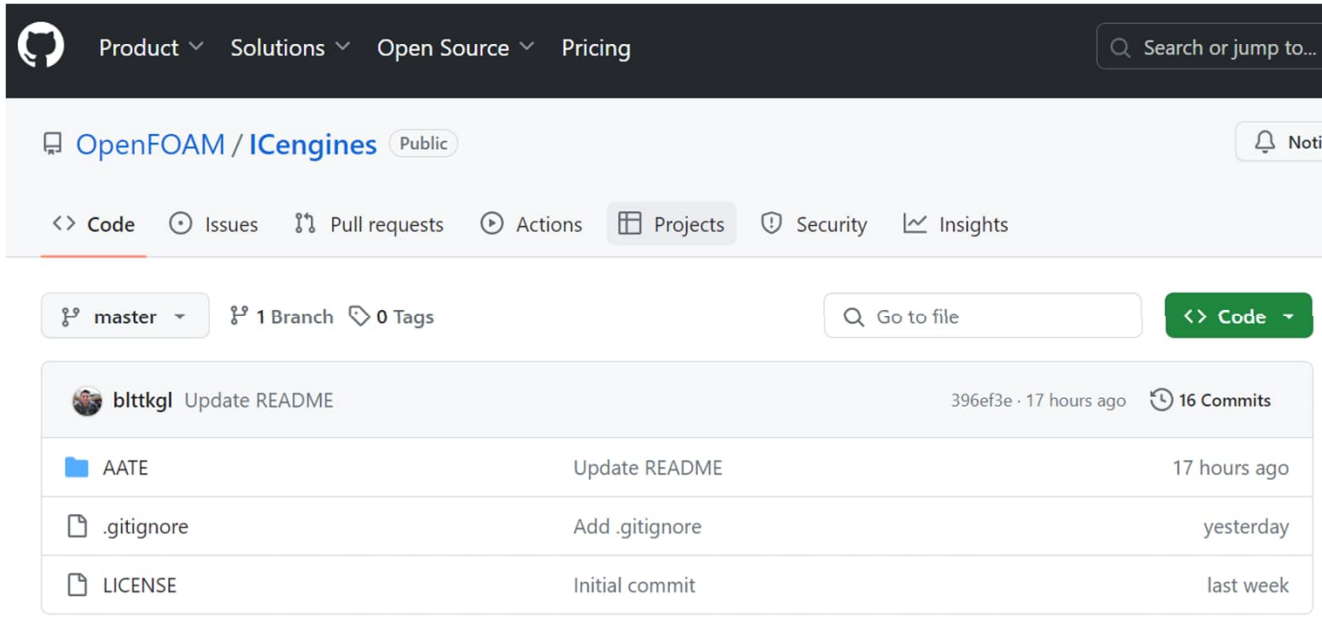
https://www.youtube.com/watch?v=EKZjcYNGCfg

Aalto University

https://primo.aalto.fi/permalink/358AALTO_INST/ha1cg5/alma999731679706526

AATE repository

- All functionality exists in OpenFOAM-dev
- AATE includes:
  - Industrial-scale case setups with meshes
    - TCC-III engine tutorial with 3 meshes
      - Two commercial mesh: coarse and fine
      - snappyHexMesh user can generate themselves*
  - Function objects for engine-specific post-processing
  - Any other engine-functionality that does not fit in OpenFOAM-dev



- Release schedule (plan)
  - OpenFOAM-dev beta version available for testing and commenting until August 2024
  - AATE v12 stable release in August 2024
  - Yearly update 1 month after stable OpenFOAM releases

https://github.com/OpenFOAM/ICengines

* Still under development

Outcome, future possibilities

- We are hoping to collaborate further on:
  - Further multi-region / CHT developments
  - Combustion modelling (Multi-mode aspects, TCI modelling, combustion concepts)
  - AMR (e.g. runtime load-balancing, refinement level field transferral to new mesh)
  - Layer addition-removal for piston motion
  - Any other ideas?

bulut.tekgul@wartsila.com
eric.lendormy@wartsila.com