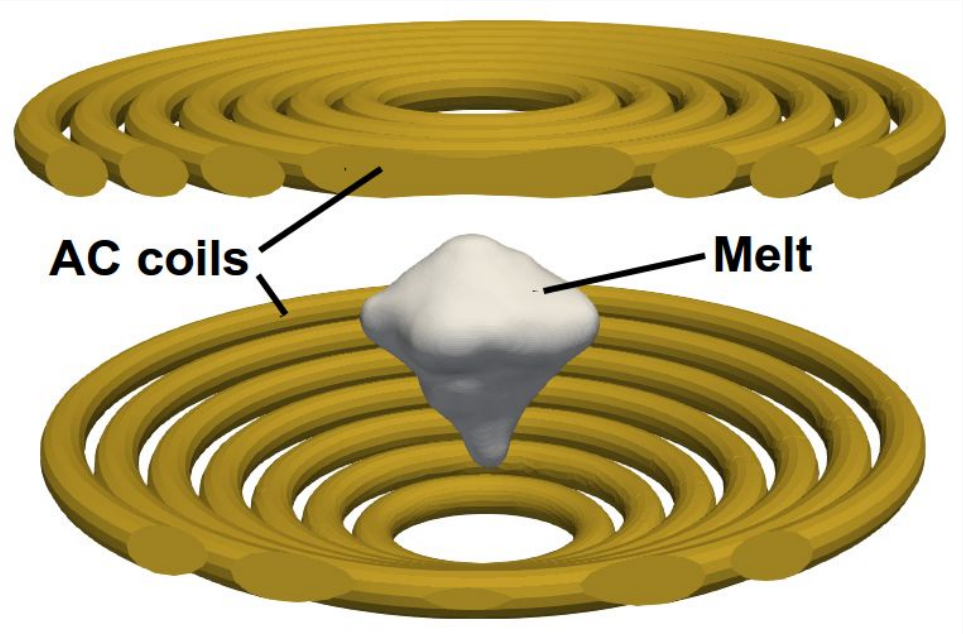
EOF-Library

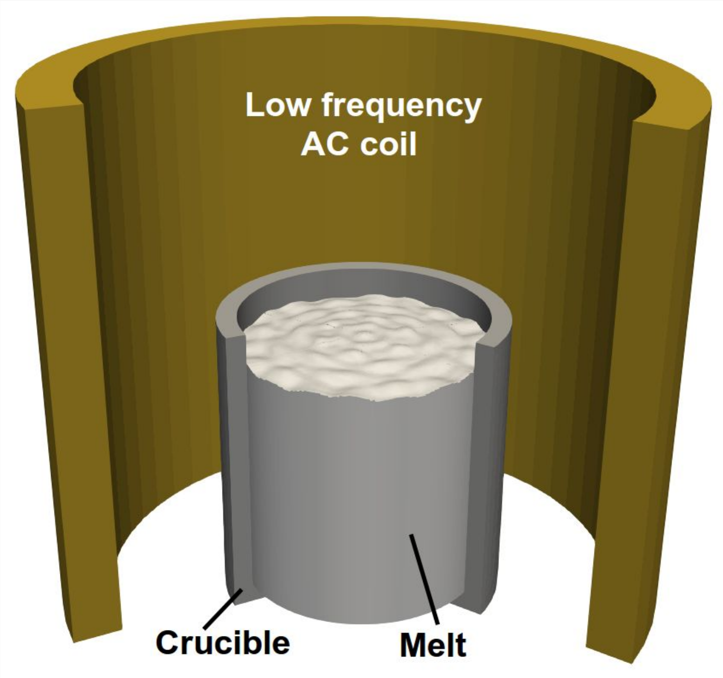
Coupler for Elmer FEM and OpenFOAM

Motivation

The first application for EOF - magnetohydrodynamics



Electromagnetic levitation melting



Surface wave generation using low frequency EM field

Requirements for MHD modelling

Fluid dynamics

- High Reynolds turbulence models
- Volume of Fluid (VOF) for free surface modelling
- Viscosity dependence on temperature

Shared (Fluid dynamics and Electromagnetics)

- 3D, transient
- Complex geometries, multi-regions
- Advanced pre- and post-processing tools
- Parallelization with good scaling
- User community, documentation

Electromagnetics

- Complex A−V magnetic vector potential formulation
- Conductivity dependence on temperature
- VxB

The right software for this task

Fluid dynamics

OpenFOAM

- *Finite Volume Method (FVM)*
- *Most popular open-source software for CFD*

Two-way coupling

EOF-Library

- *Message Passing Interface (MPI)*
- *Interpolation of internal fields*
- *Fast, robust & physics invariant*

Electromagnetics

Elmer FEM

- Finite Element Method (FEM)
- Parallelizable, linear & non-linear iterative solvers

The right way to transfer data

File based

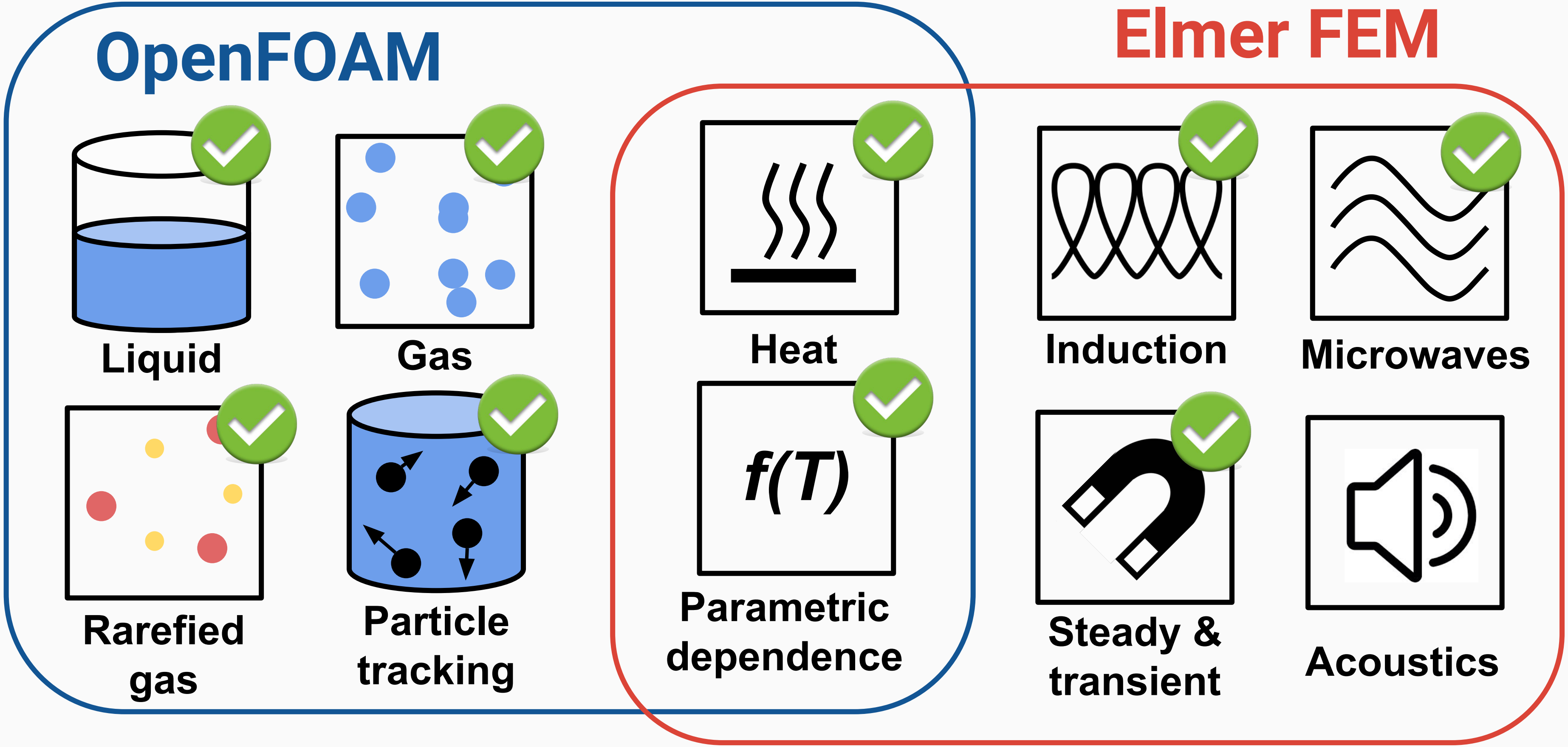
- SLOW and does not SCALE on parallel computers

MPI (Message Passing Interface) based

- Both codes already use MPI for parallelization
- Probably the most EFFICIENT coupling solution

Applications

Solvers suitable for coupling (✓ - tested)



Metallurgy

Microwave heating

Electrical devices

Plasma physics

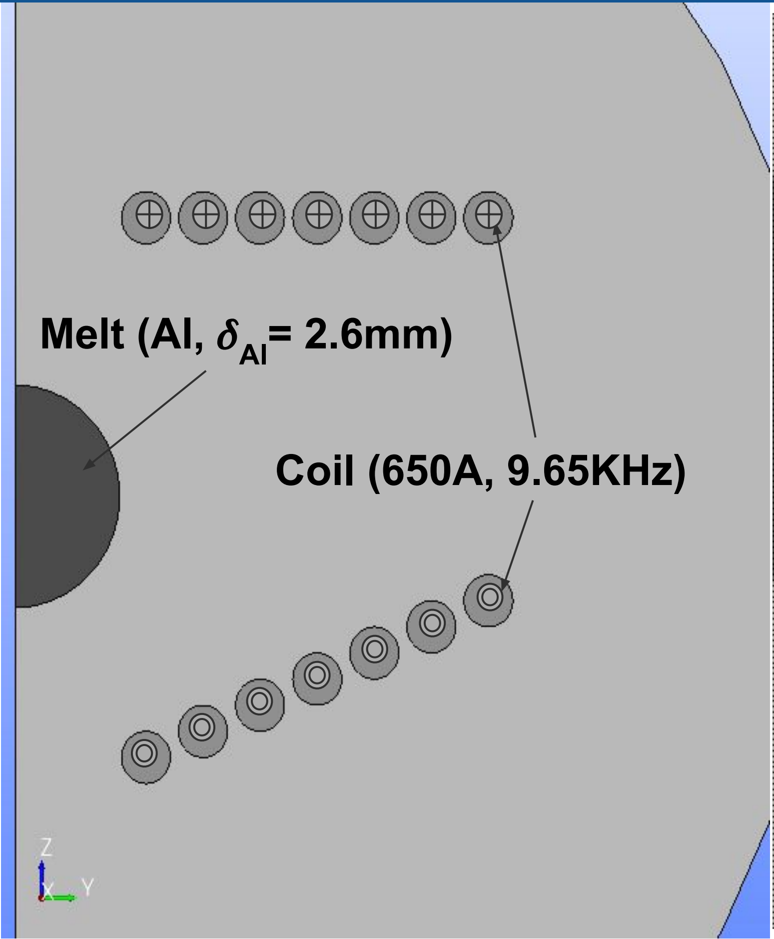
Simulated problem - EM levitation melting

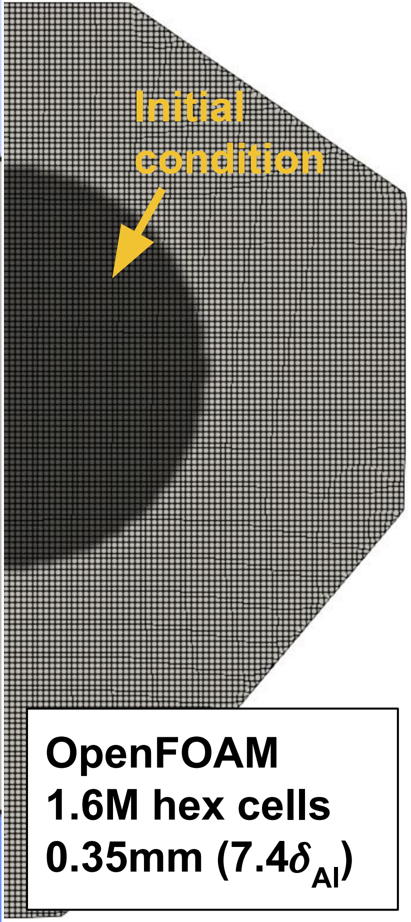
Device for electromagnetic levitation melting experiment

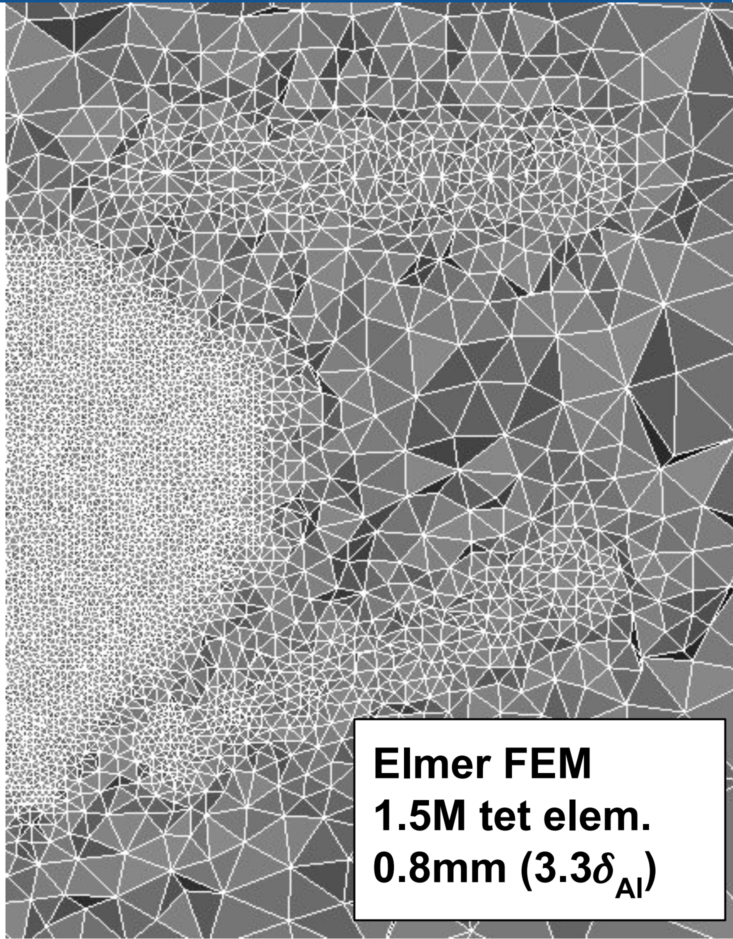
Pictures and video from

Spitans, Sergejs, et al. “Numerical Modeling of Free Surface Dynamics of Melt in an Alternate Electromagnetic Field. Part II: Conventional Electromagnetic Levitation”

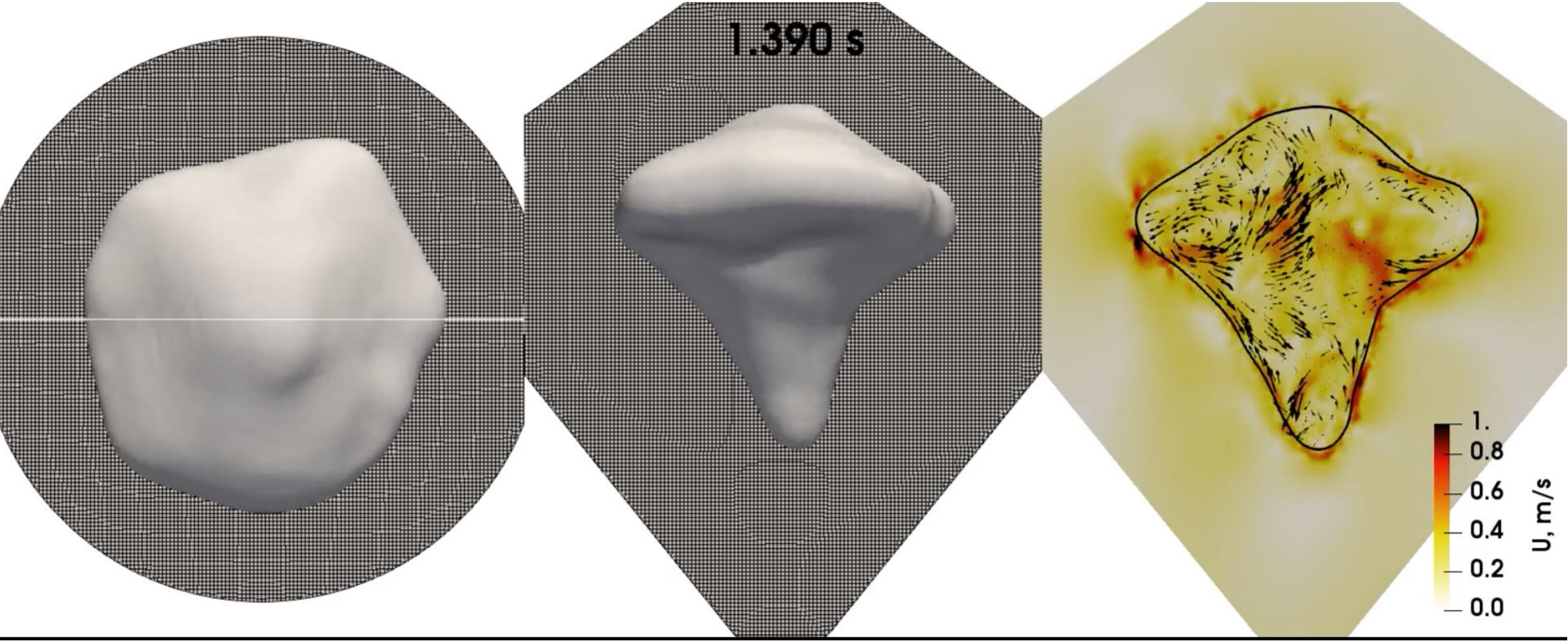
Geometry & Meshes



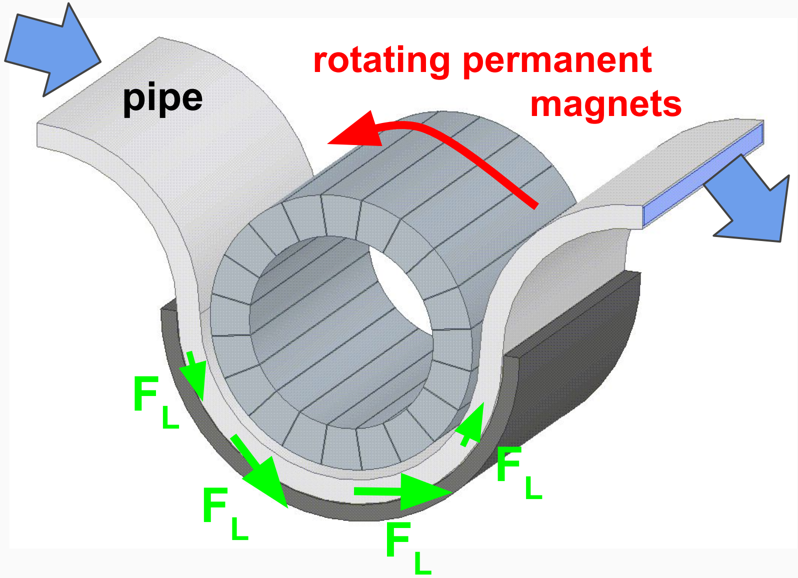


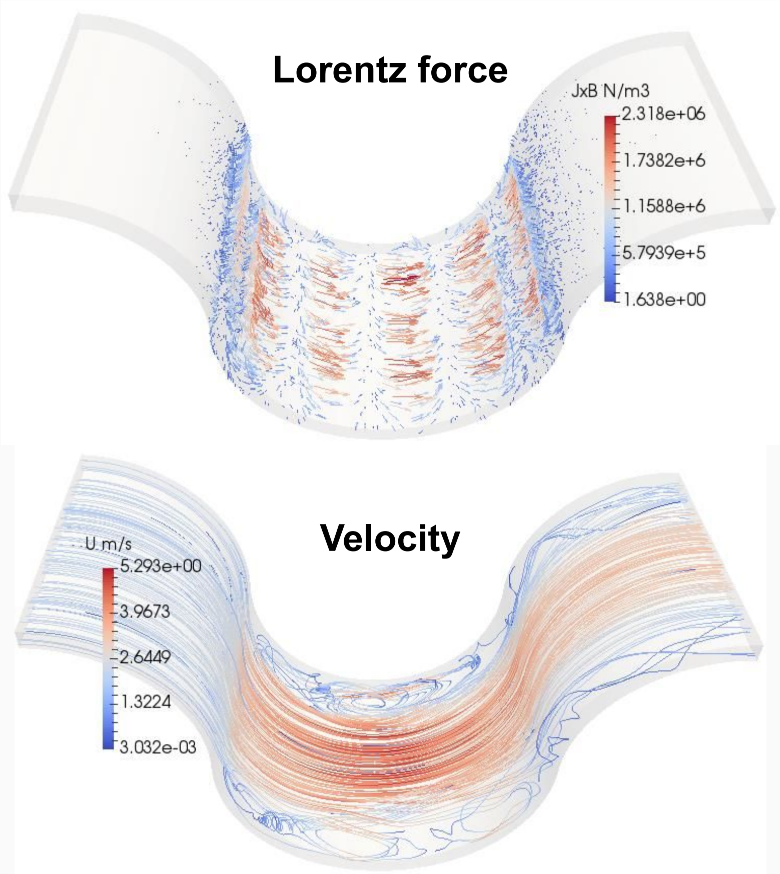


Results



Liquid metal pump

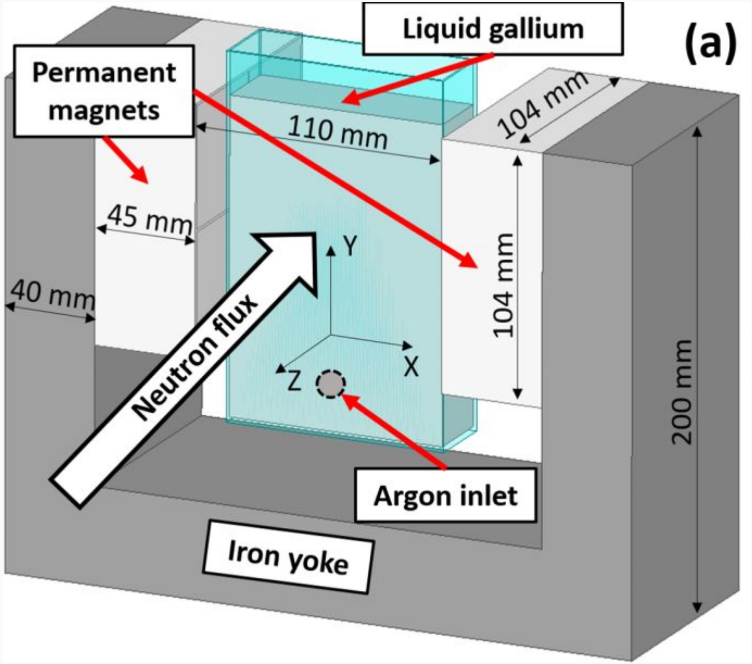




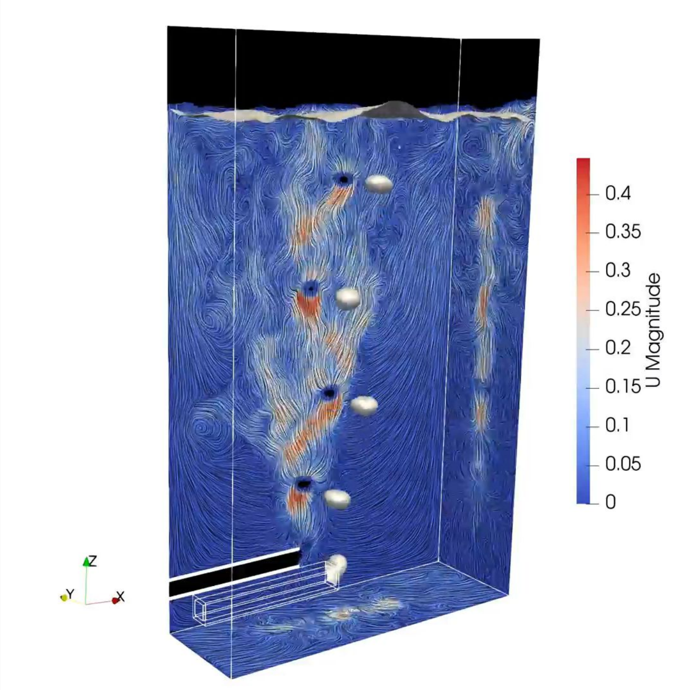
Figures from

Dzelme Valters et al. *Numerical modelling of liquid metal electromagnetic pump with rotating permanent magnets* (2018)

Bubbles in liquid metal



Birjukovs M., Dzelme V. et al. *Phase boundary dynamics of bubble flow in a thick liquid metal layer under an applied magnetic field* (2020)



Klevs M., Birjukovs M. et al. *Dynamic mode decomposition of magnetohydrodynamic bubble chain flow in a rectangular vessel* (2021)

Metallurgy

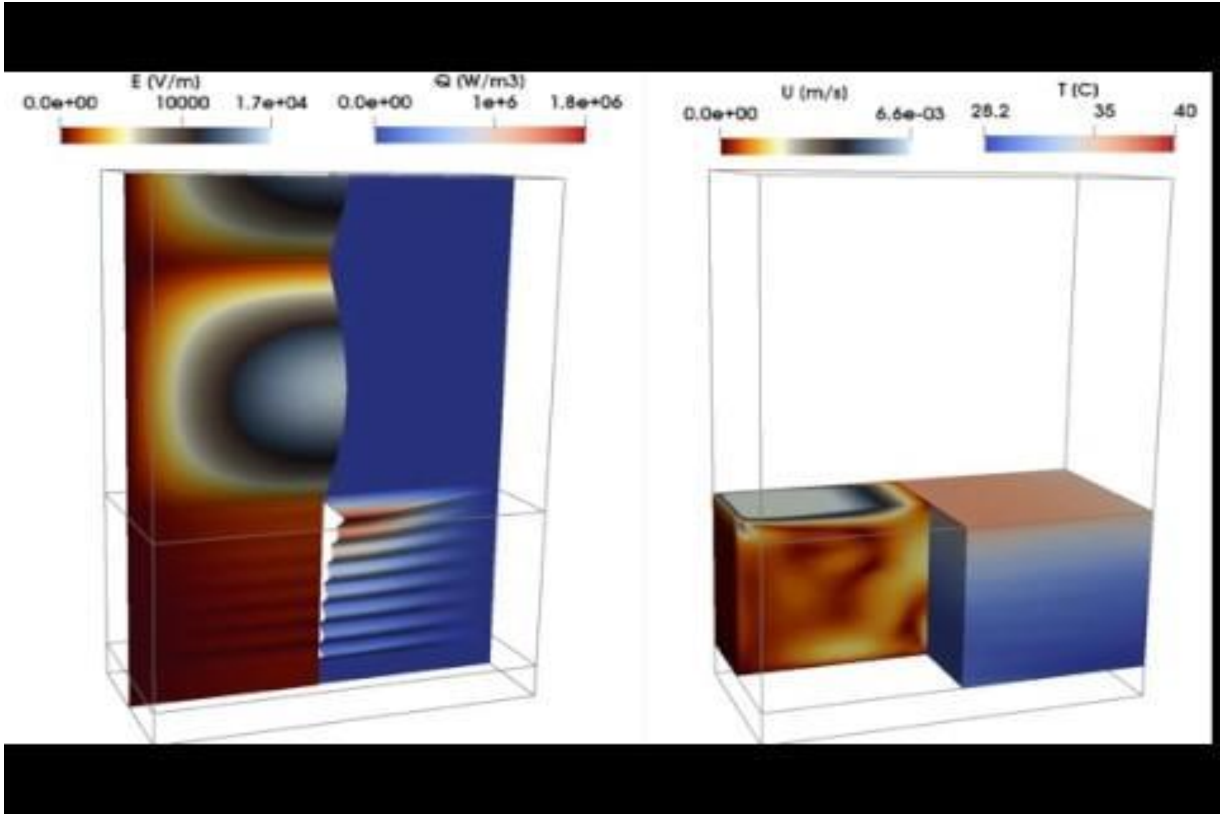
Microwave heating

Electrical devices

Plasma physics

Heating in rectangular waveguide

Figure from W. Klinbun, P. Rattanadecho. *Investigation into heat transfer and fluid flow characteristics of liquid two-layer and emulsion in microwave processing* (2016)



Vencels J., Birjukovs M., Kataja J., Råback P. *Microwave heating of water in a rectangular waveguide: validating EOF-Library against COMSOL Multiphysics and existing numerical studies (2019)*

Microwave heating of liquids

Fluid dynamics

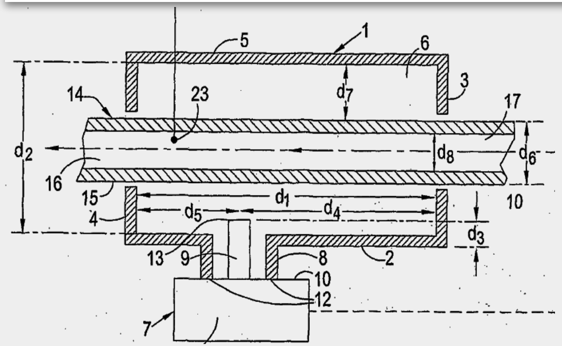
Heat transfer

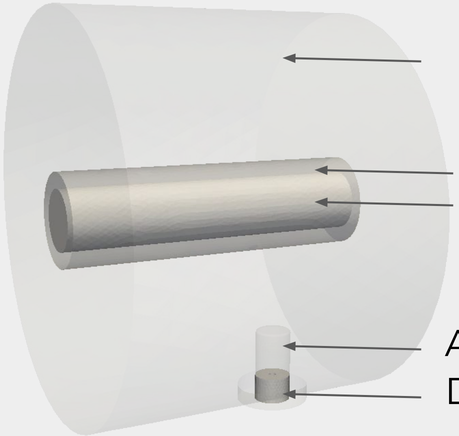
Electromagnetics

Microwaves

It is a three-way coupled problem. At first, MWs heat the fluid, and flow transfers the heat mostly in a convective way. Change in temperature affects liquid properties that, in turn, alter flow characteristics and MW propagation.

Zadyraka et al. Method for treating a fluid with microwave radiation





MW analysis

Two common types of MW inlet ports

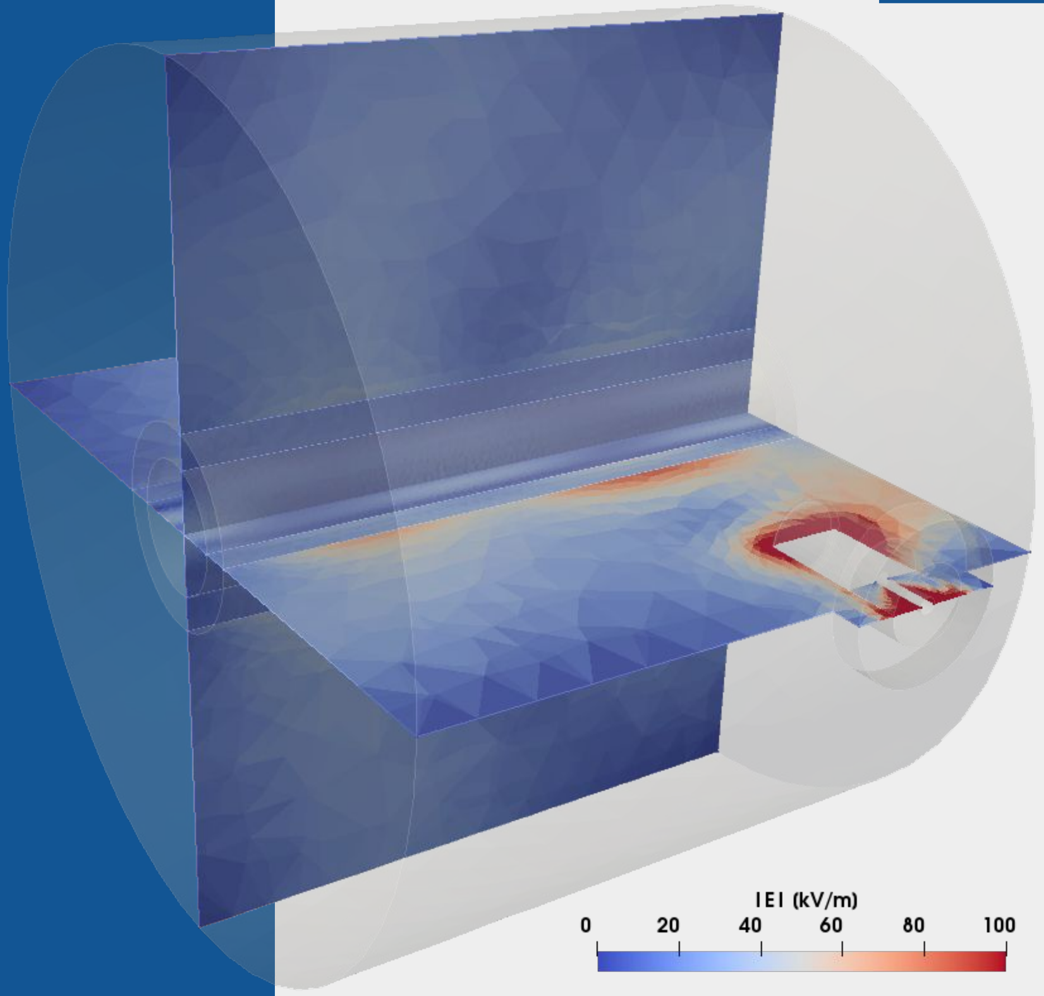
- Dipole antenna
- Rectangular waveguide

Permittivity is the most important parameter involved in electromagnetic wave propagation and dissipation.

For a fixed frequency (e.g. 2.45 GHz) temperature-dependent permittivity given as an complex function $\varepsilon_{Re}(T)$, $\varepsilon_{Im}(T)$

Numerical models include two MW heating mechanisms

- Dielectric (polarization)
- Conductive



Computational fluid dynamics (CFD)

Heat transfer in fluids and gases takes place in conductive and convective ways. Turbulent flow has an additional convective term that increases the efficiency of the overall heat transfer process.

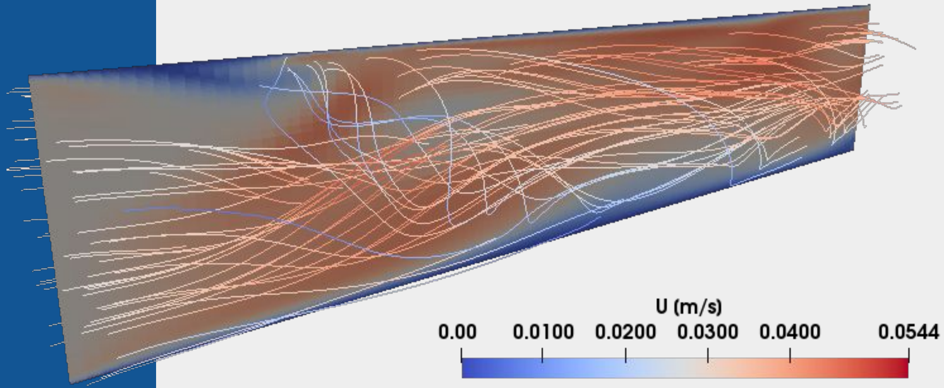
Heat transfer in solid materials, evaporation, and multiphase (gas + liquid) models can be added too.

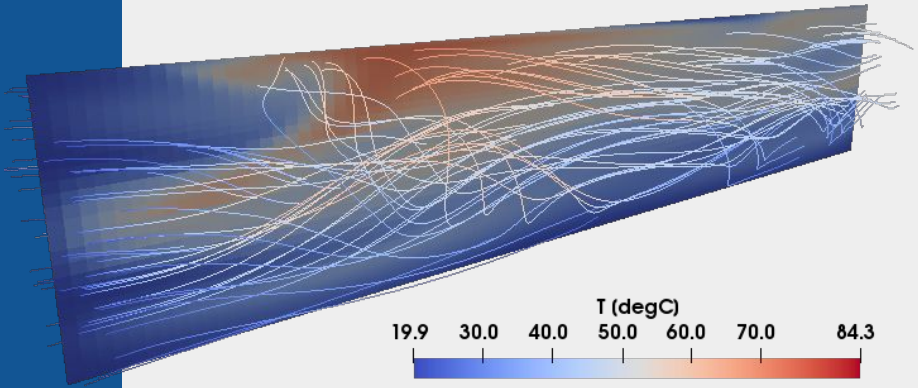
Fluids typically have three main temperature-dependent properties that affect numerical results

- Viscosity
- Density
- Thermal conductivity

Finally, there are different viscosity models

- Newtonian (water, alcohol)
- Non-Newtonian (ketchup, resin)





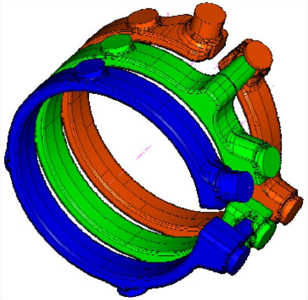
Metallurgy

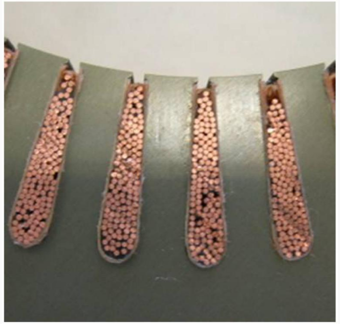
Microwave heating

Electrical devices

Plasma physics

Cooling of permanent magnet motor





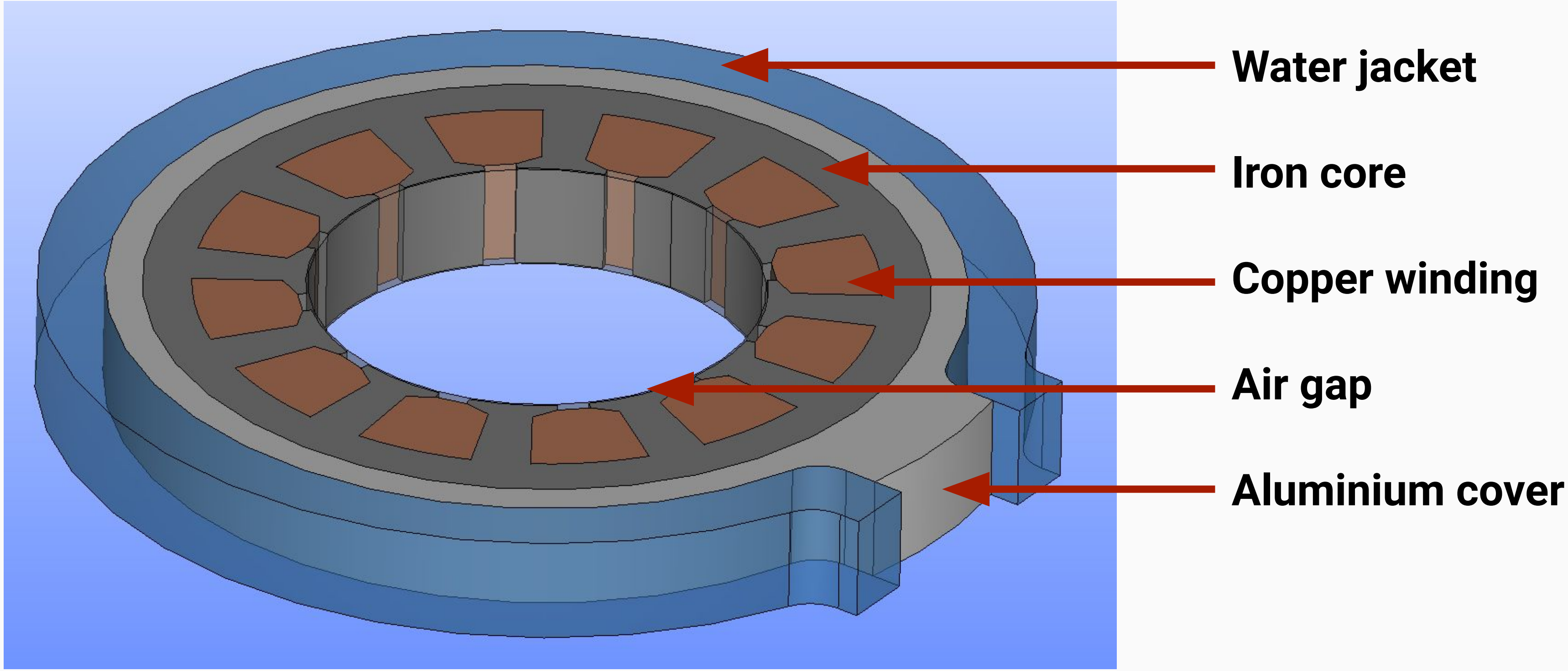
Figures of motor from: Sato, Y., Ishikawa, S., Okubo, T., Abe, M., & Tamai, K. (2011). Development of High Response Motor and Inverter System for the Nissan LEAF Electric Vehicle

$\rho(20\ ^{o}C) = 1.7 \times 10^{-8}\ (\Omega\ m)$

$\rho(80\ ^{o}C) = 2.1 \times 10^{-8}\ (\Omega\ m)$

60 ^{o}C higher temperature
~24% increase in losses!

Cooling of permanent magnet motor



Janne Keränen (Elmer FEM model)
Aku Karvinen (OpenFOAM support)

Cooling of permanent magnet motor

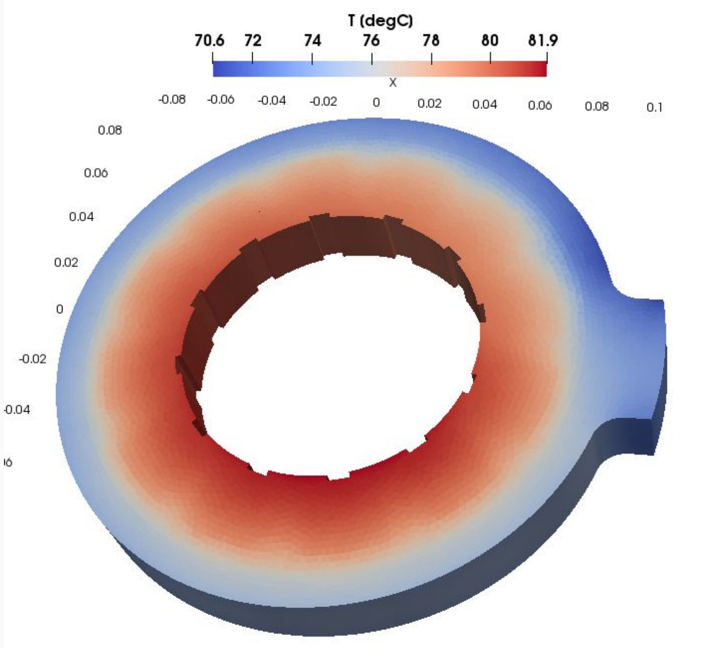
Assumptions:
Steady state

Constant inlet water temperature and flow rate

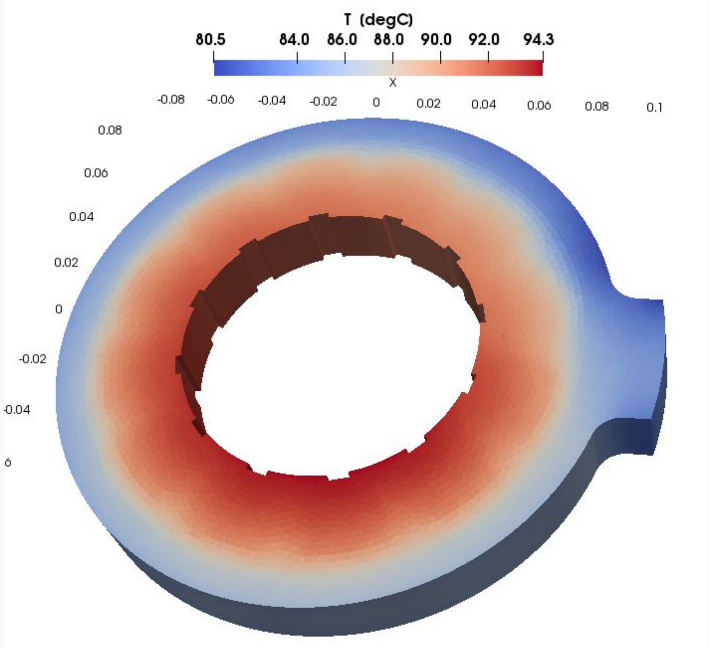
Homogeneous and constant current density in windings

Conclusion:
Preliminary results show importance of ρ(T) in simulation models

Temperature distribution in solid regions



ρ = const



ρ(T)

Metallurgy

Microwave heating

Electrical devices

Plasma physics

Consulting business in Czechia

PlasmaSolve

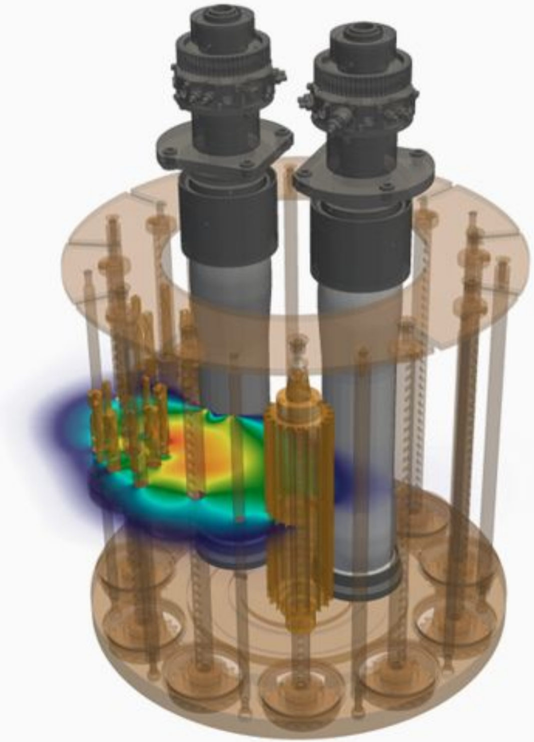
HOME NEWS PVD/PECVD SIMULATION ˅ NEWSPACE ˅ CROSS-SECTIONAL SCIENCE ˅ ABOUT US CONTACT

We design and optimize plasma-powered processes and equipment

Our simulation helps you visualize and comprehend the process like never before and find the best engineering solution to your challenge.

SEE OUR FIELDS OF EXPERTISE

learn how we work or simply get in touch



How it works

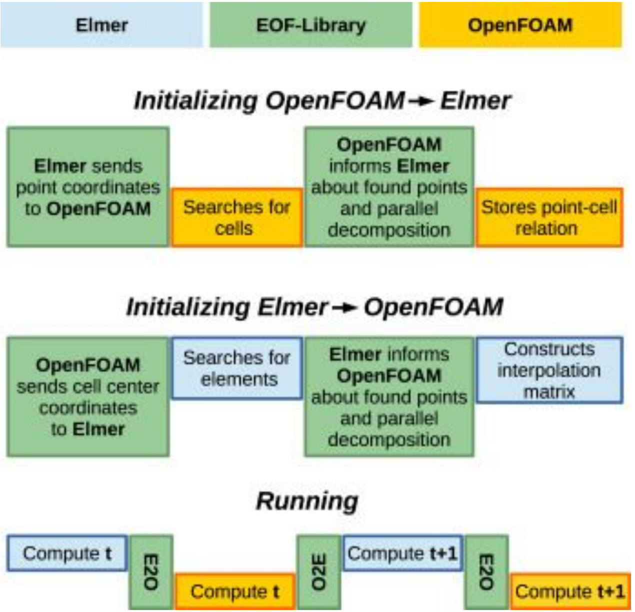
Coupling scheme

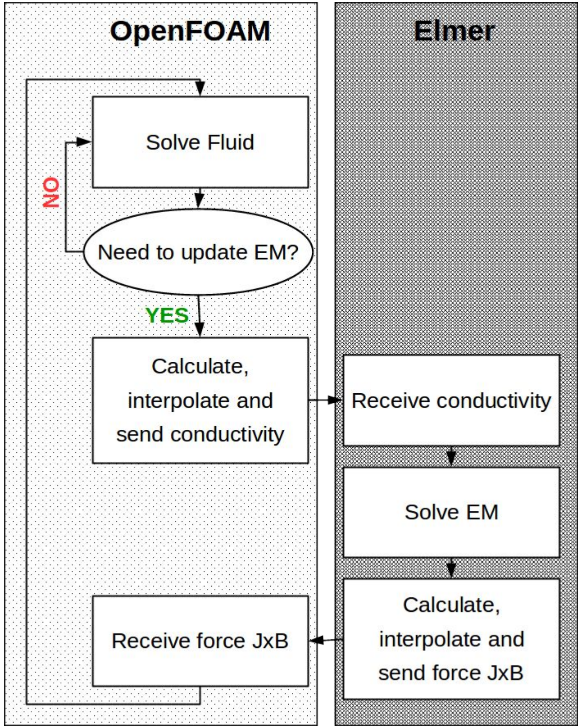
Coupler's OpenFOAM part

```
├── coupleElmer
│   ├── Elmer.C
│   ├── Elmer.H
│   └── Make
│       ├── files
│       └── options
```

Coupler's Elmer part

```
├── Elmer2OpenFOAM
│   └── Elmer2OpenFOAM.F90
└── OpenFOAM2Elmer
    └── OpenFOAM2Elmer.F90
```





In the code

Interpolating and sending fields from Elmer to OpenFOAM

Checks whether Elmer sends message,
if not then sleeps 10ms (saves CPU resources)

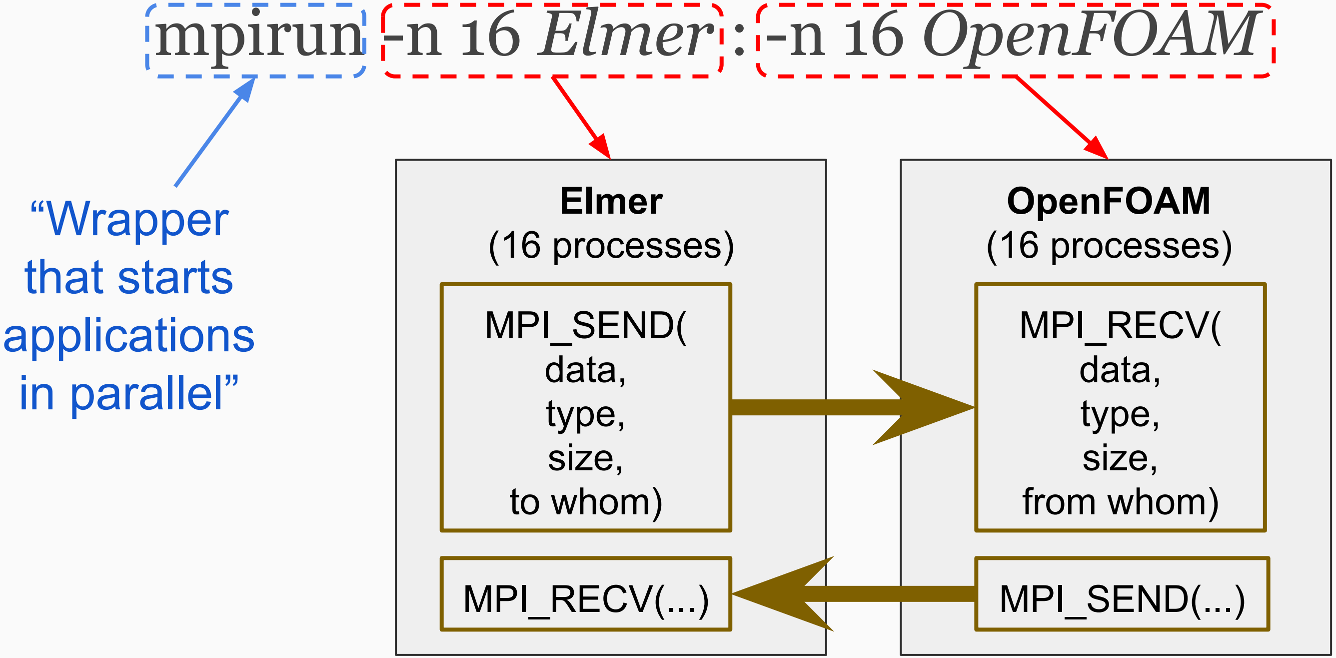
```
void Foam::Elmer::recvScalar(volScalarField& field)
{
    int i, j, flag;

    Info<< "Receiving scalar field from Elmer.." << endl;

    for ( i=0; i<totElmerRanks; i++ ) {
        if ( ELp[i].nFoundCells > 0 ) {
            while ( true ) {
                MPI_Iprobe(ELp[i].globalRank, 1000, MPI_COMM_WORLD, &flag, MPI_STATUS_IGNORE);
                if (flag) break;
                nanosleep((const struct timespec[]){{0, 10000000L}}, NULL);
            }
            MPI_Recv(ELp[i].recvBuffer0, ELp[i].nFoundCells, MPI_DOUBLE, ELp[i].globalRank,
                     1000, MPI_COMM_WORLD, MPI_STATUS_IGNORE);
            for (j=0; j<ELp[i].nFoundCells; j++ ) {
                field[ELp[i].foundCellsIndx[j]] = ELp[i].recvBuffer0[j];
            }
        }
    }
}
```

Projector matrix (interpolation) is applied on variable

```
! Send fields
DO j=1,nVars
  VarName = ListGetString( Params, 'Target Variable '//TRIM(I2S(j)), Found )
  Var => VariableGet( CurrentModel % Mesh % Variables, VarName )
  IF(.NOT. ASSOCIATED( Var ) ) THEN
    CALL Fatal('Elmer2OpenFOAMSolver','Variable '//TRIM(VarName)//' does not exist in Elmer mesh!')
  END IF

  DO i = 0, totOFRanks - 1
    IF ( OFp(i) % nFoundCells > 0 ) THEN
      OFp(i) % OFVar % Values = 0
      CALL CRS_ApplyProjector( OFp(i) % OFMesh % Projector % Matrix, Var % Values, &
                               Var % Perm, OFp(i) % OFVar % Values, OFp(i) % OFVar % Perm )

      CALL MPI_ISEND( OFp(i) % OFVar % Values, OFp(i) % nFoundCells, MPI_DOUBLE, &
                      OFp(i) % globalRank, 1000, MPI_COMM_WORLD, OFp(i) % reqSend, ierr)
    END IF
  END DO

  DO i = 0, totOFRanks - 1
    IF ( OFp(i) % nFoundCells > 0 ) THEN
      CALL MPI_TEST_SLEEP(OFp(i) % reqSend, ierr)
    END IF
  END DO
END DO
```

MPI - Message Passing Interface

Running on 16 cores:



Accuracy & Performance

Interpolation test - **unit cube**

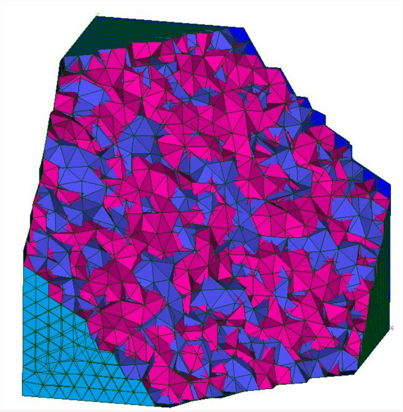
Elmer & OpenFOAM meshes are different

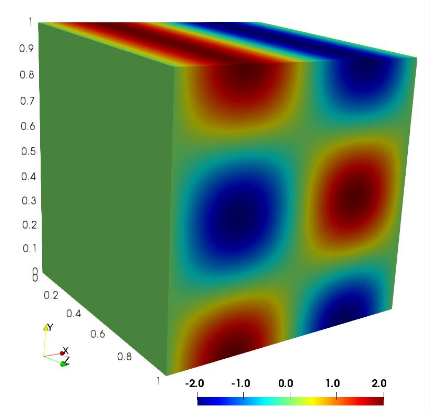
Refinement level

- 0.1 (4.7k tets)
- 0.05 (32k tets)
- 0.025 (245k tets)

Reference scalar field

2 sin[2πx] cos[2πy]





Elmer-to-OpenFOAM			
Tet size	**0.1**	**0.05**	**0.025**
Relative Error %	8.0	2.3	0.57

OpenFOAM-to-Elmer			
Tet size	**0.1**	**0.05**	**0.025**
Relative Error %	11	4.0	1.5

Strong scaling test

13.1M cells / elements

				Initialization		Runtime	
Cores	**Mem,GB**	**Wall,s**	**Wall eff**	**O2E,s**	**E2O,s**	**O2E,s**	**E2O,s**
4	25.9	5837	100%	175	96	10	0.36
16	28.5	1500	97%	36	29	2.3	0.13
64	29.4*	420	87%	12	9.3	0.65	0.1
256	77.7*	173	53%	4.4	4.9	0.22	0.14

Conclusion:

Coupler makes a negligible performance overhead comparing to computation time

*estimated (not accurate)

How to get started

Do-It-Yourself way for learning "EOF"

1) **Start with Elmer and OpenFOAM**
    a) Documentation
    b) Tutorials
    c) Following examples
2) **Learn how to use Linux** - commands, Docker, compilation
3) **Follow EOF-Library examples**
    a) Test case - levitation2D
    b) OpenFOAM solver - mhdInterFoam
    c) Article - *EOF-Library: Open-source Elmer FEM and OpenFOAM coupler for electromagnetics and fluid dynamics*

EOF Consulting

- *Research (physics & optimization)*
- *Simulation workflow development*
- *Training & support*

vencels@eof-consulting.com
+371 24786762

CEO - Juris Vencels


DEMO