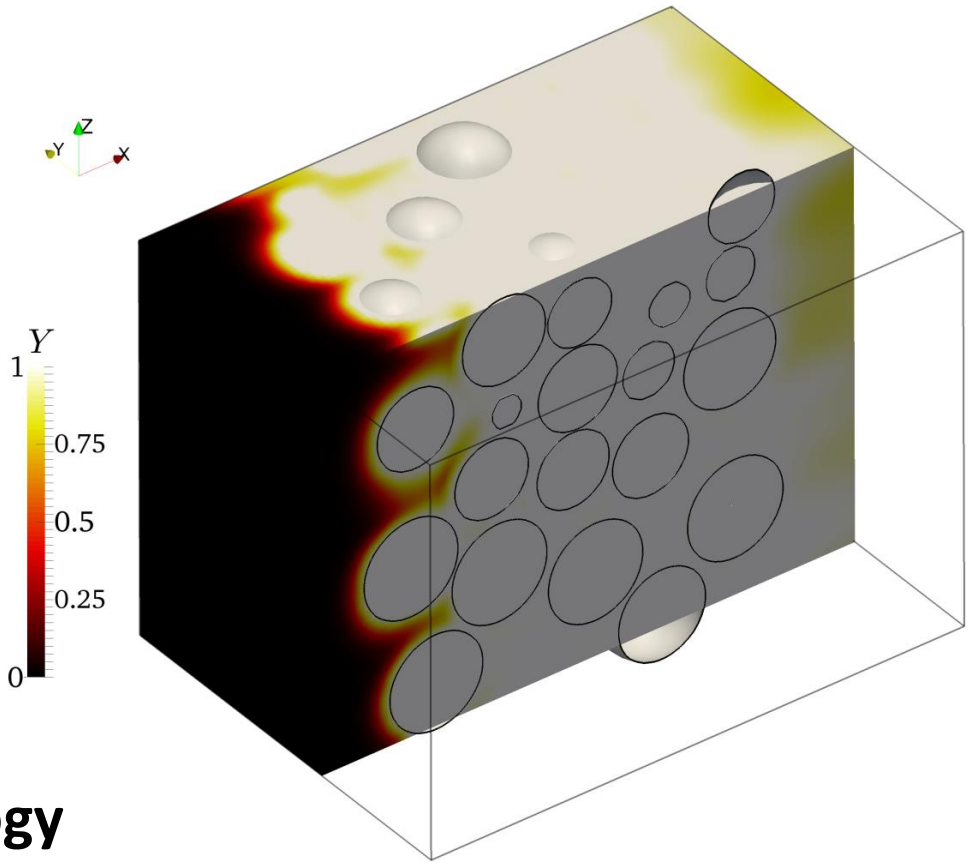
Perspectives for

Modeling Reactive Particulate Systems using Open-Source Tools

Stefan Radl,
Federico Municchi,
and Thomas Forgber

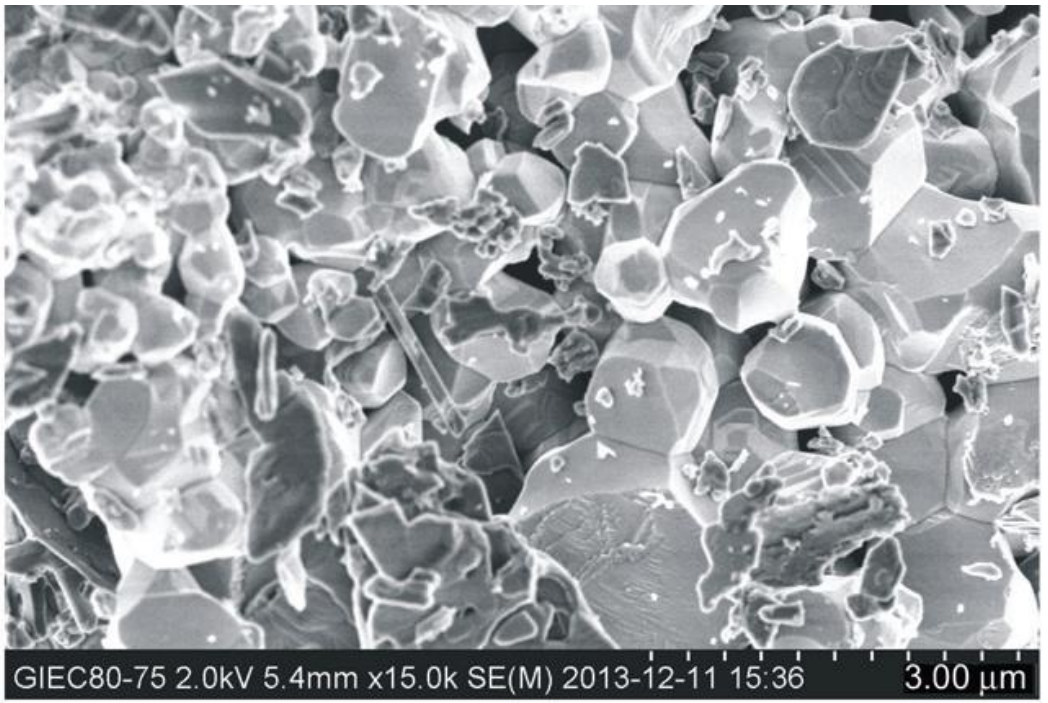


Graz University of Technology

Why Modeling Reactive Flows?

1 - Next Generation Chemical Looping Processes

- Methan reforming via **partial oxidation** using solid oxygen carriers has significant economic advantages.
- **Iron-based oxygen carriers**, nano-structured and doped with a second metal, are promising.
- Rational **design** is challenging (sintering, optimal pore structure) and **requires a model**.

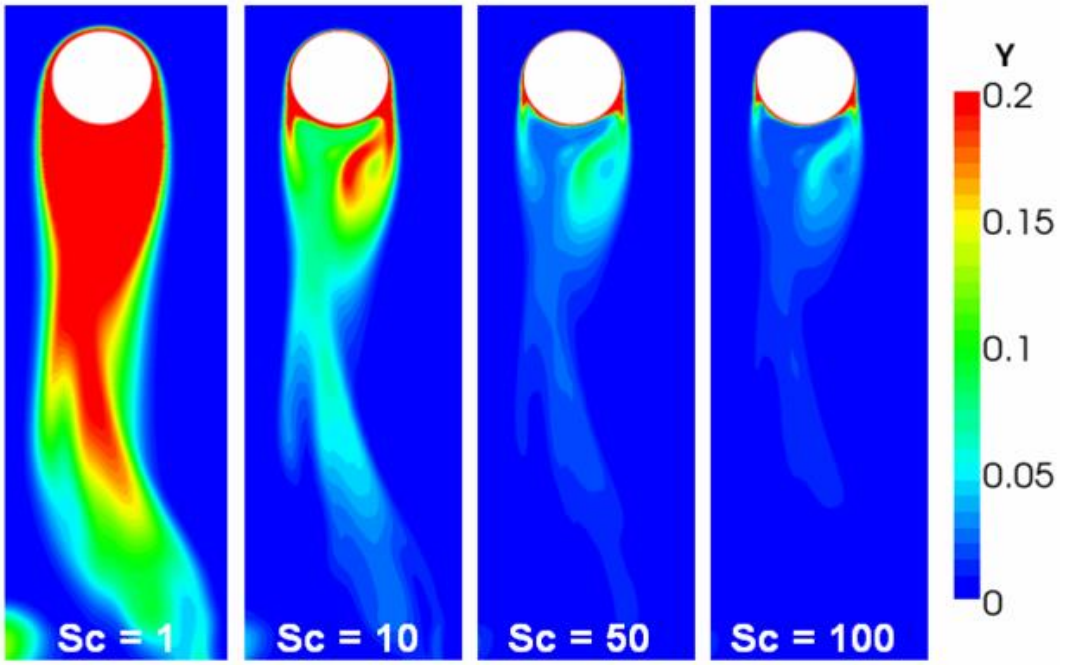


SEM image of a typical nano-structured Ni-modified Fe_2O_3/Al_2O_3 oxygen carrier (Wei et al., 2014).

Why Modeling Reactive Flows?

2 - Scientific Curiosity

- Resolution of **flow, concentration, and temperature** field by solving the relevant transport equations.
- **Detailed information** allows (i) profound process **understanding**, and (ii) is the basis for **coarse-grained** models (e.g., DNS → LES)

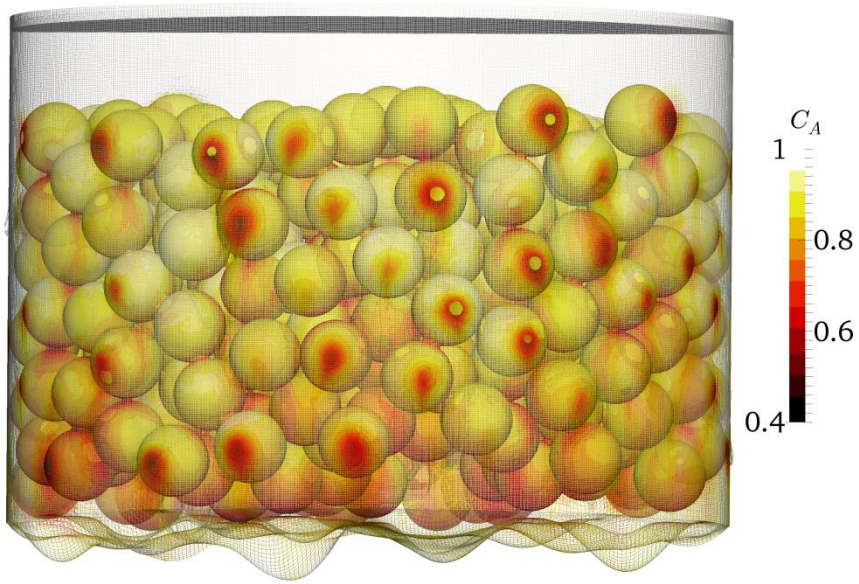


Instantaneous concentration fields in the wake of a single particle (Re = 500).

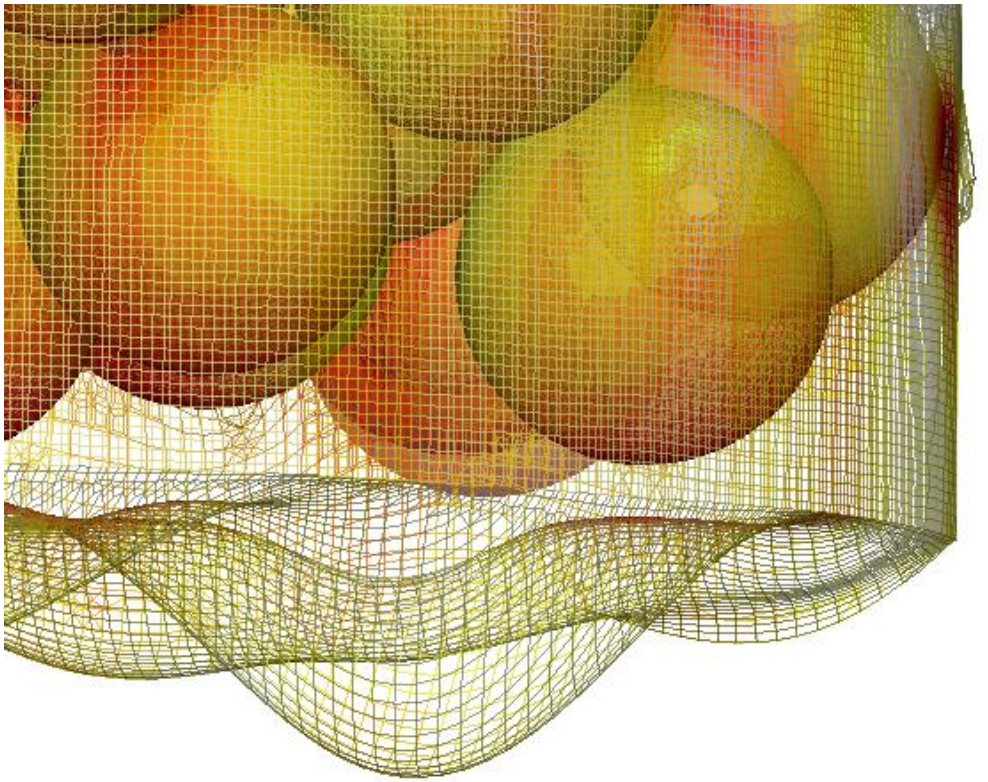
Why Modeling Reactive Flows?

2 - Scientific Curiosity

- Modeling on **multiple levels** (co-current or sequential) and **exploitation of "big data"** is key.



Concentration field in liquid catalytic reactor
(Re ~ 1, Sc ~ 500,
Master Thesis Rafael Pichler, 2015).



Which Modeling Tools Exist?

1 - Multipurpose Continuum Solvers

- Matlab /octave / Julia
- OpenFOAM®
- CFDEM®
- ANSYS Fluent
- AVL Fire®

2 - Multipurpose Lagrangian Solvers

- Yade / WooDEM
- LAMMPS / LIGGGHTS®
- SimPARTIX®
- EDEM
- Itasca PFC

Open-source

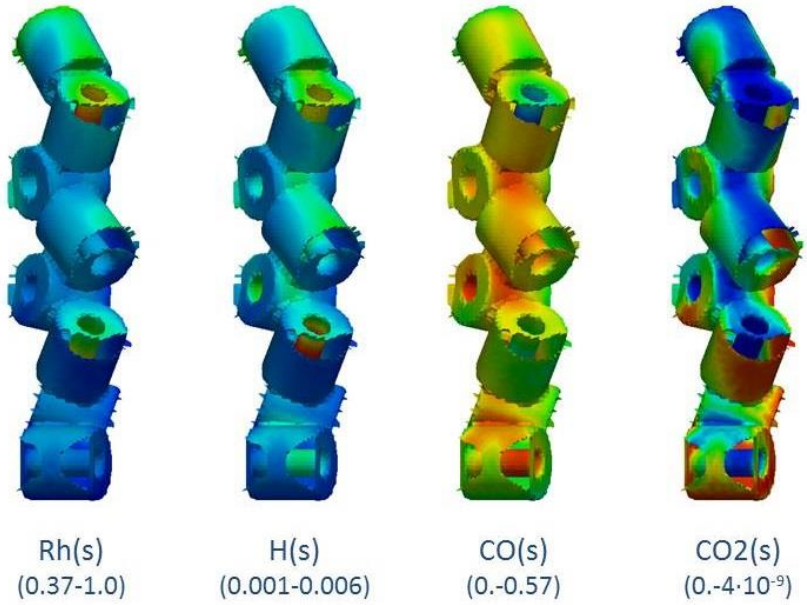
Open Issues

1. **Sub-Particle** scale modeling (first step: OpenFOAM®'s „pyrolysisModel")
2. Advanced **data analysis** tools in parallel (first step: swak4Foam)
3. Efficient **parallelization** for Euler-Lagrange co-simulations
4. Advanced **coupling models** (i.e., for the fluid-particle interaction force)

Which Modeling Tools Exist?

3 - Kinetics & Transport Tools

- CANTERA
- OpenFOAM® / CatalyticFoam
- DETCHEM / DC4OpenFOAM
- CHEMKIN



Adsorbed species mass fraction distribution on the surface of non-porous particles
http://www.catalyticfoam.polimi.it

Open Issues

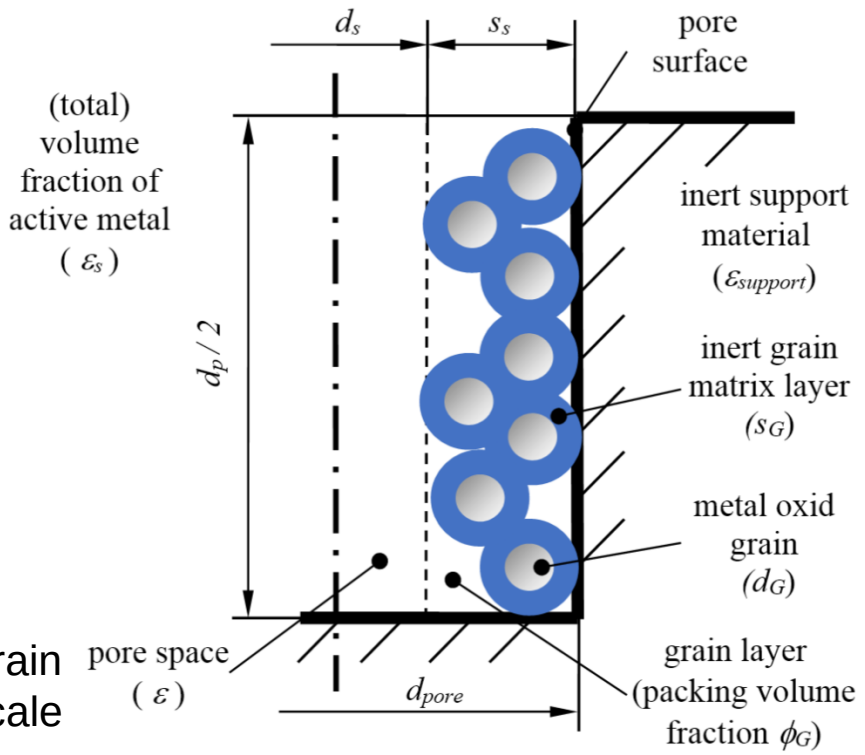
1. **Coupling** to high-performace CFD solvers missing
2. Limited to 1D transport in a **single particle**
3. Particles cannot **move**
4. Transport in **porous particles** missing

Open-source

What is ParScale?

A Compilation of Particle Scale Models

- Model transport and reactions in **porous particles** with **“grains”**
- Designed for **stiff systems** (CVODE as integrator)
- ParScale can manage **large particle clouds** with O(10^6) particles in parallel via LIGGGHTS®-based data containers

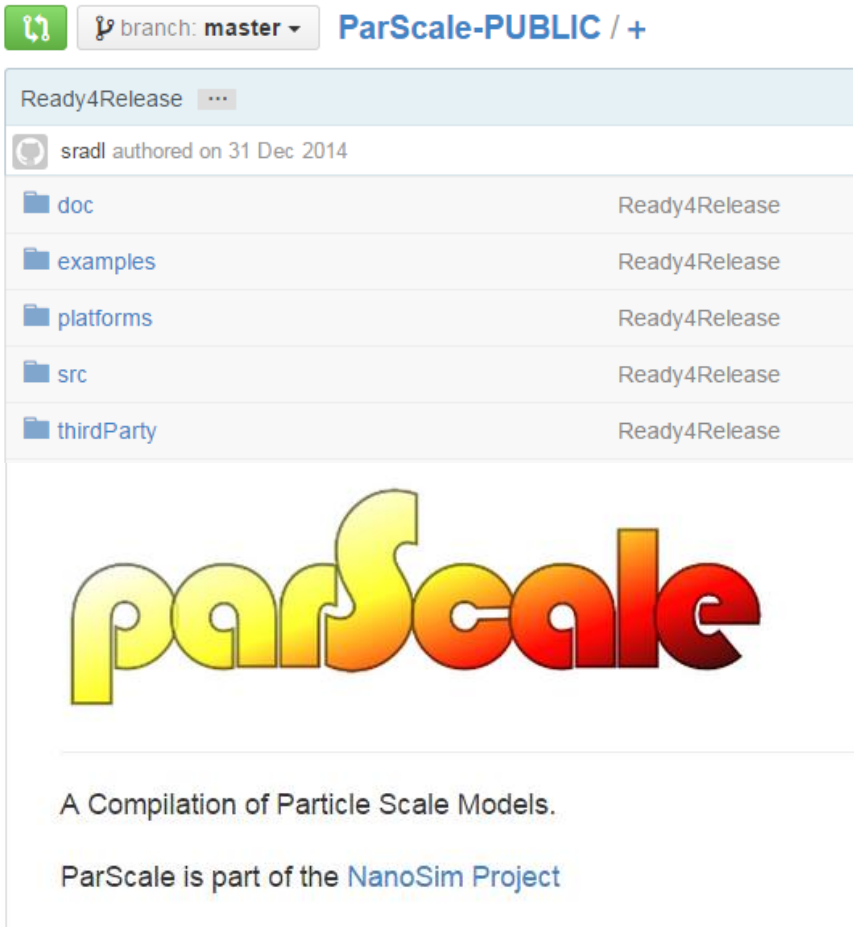


Typical pore and grain model available in ParScale

What is ParScale?

A Compilation of Particle Scale Models

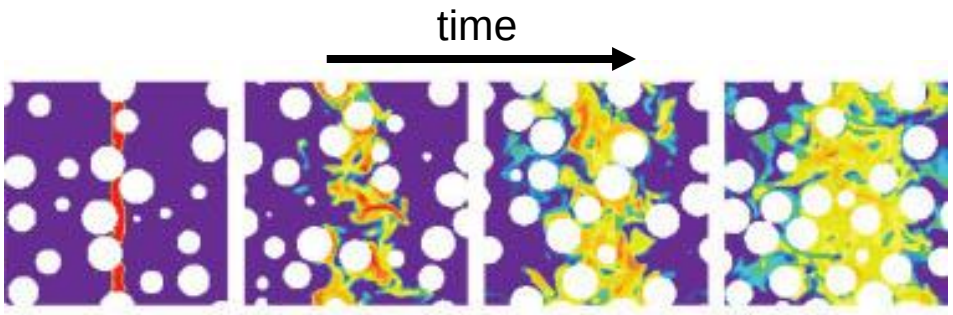
- Designed to run in **standalone mode**, or to be linked with **LIGGGHTS®**
- **Flexible** (Matlab-like input script; arbitrary number of species, CHEMKIN file format interface)
- Licensed under **LGPL**
- https://github.com/CFDEMproject/ParScale-PUBLIC



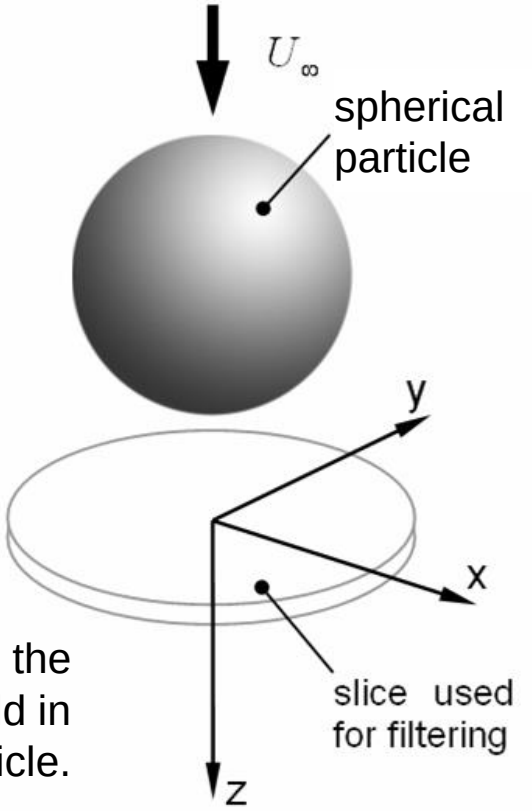
What is Filtering?

A Spatial Data Analysis Strategy

- **Filtering = Spatial averaging** of **microscopic quantities**, e.g., forces, concentration & temperature fields, reaction rates to obtain **effective quantities.**
- Example: Measure **Dispersion Coefficients**



Dispersion of a concentration pulse (Derksen, 2008).



Sketch of the setup used for the filtering of the concentration field in the wake of a particle.

What is Filtering?

The Filtering Toolbox CPPPO

- Spatial averaging **across multiple processors**
- Designed as **library** to be linked to **OpenFOAM®**
- **CSV interface** (e.g., ANSYS Fluent output)
- **Flexible** (Matlab-like input script; filtering, sampling and binning operations configured at runtime)
- Licensed under **LGPL**

branch: C3PO_dev ▾ CFDEMcoupling-RADL / src / c3po / +

This branch is 31 commits ahead, 13 commits behind master

Major Update of sampling ···

sradl1981 authored 13 hours ago

..

applications	Merge master in C3PO_dev
core	Major Update of sampling
doc	Major Update of sampling
etc	Removed bugs
examples	Major Update of sampling
interface_CSV	Removed bugs
interface_OF	Update

CPPPO

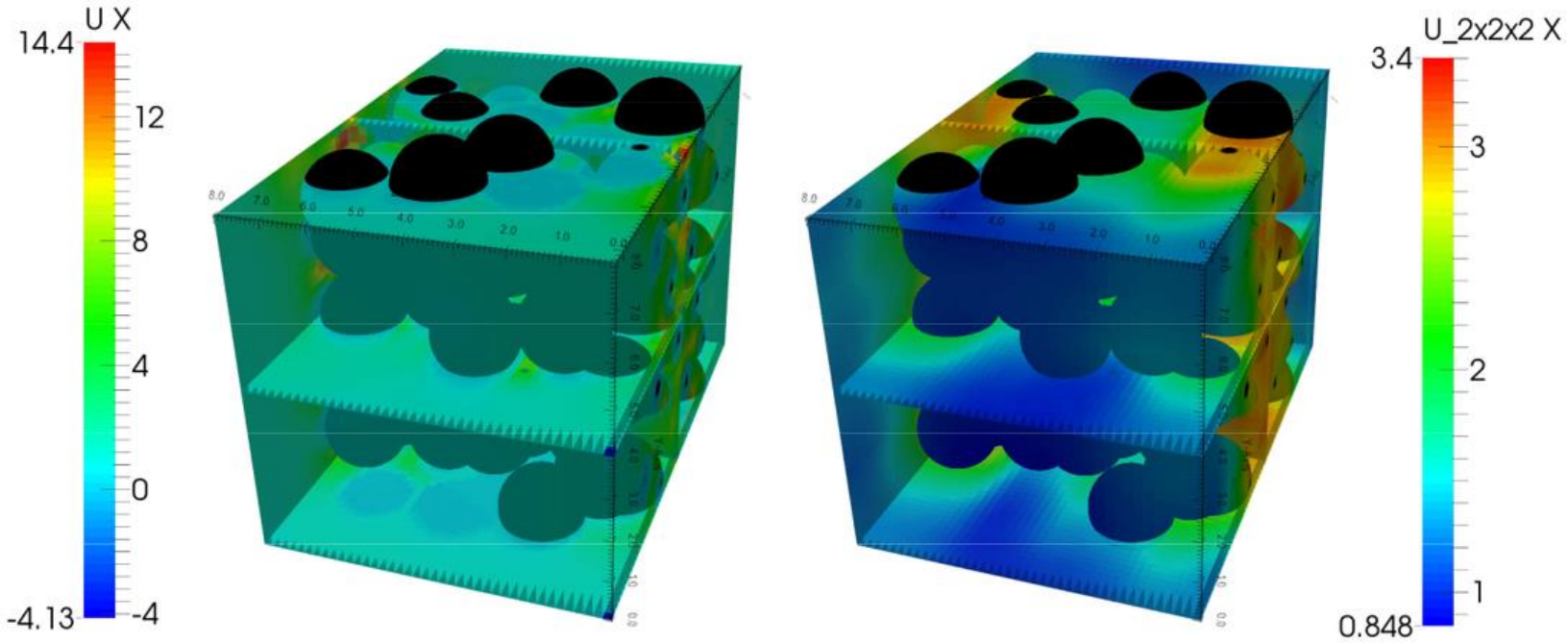
A Compilation of Fluid-Particle Post Processing routines.

CPPPO is part of the NanoSim Project

What is Filtering?

Spatial Data Analysis – An Example

- **Direct numerical simulation** of flow around a random particle ensemble
- On-the-fly top-hat filtering during a CFDEM® run using CPPPO



Conclusion

IPPT uses CFDEM® as the main Euler-Lagrange(-Euler) simulation engine

- Based on **OpenFOAM®**
- Parallel efficient **up to ~ 128 cores**
- Improved models & algorithms (e.g., coupling, forces, sprays, etc.)
- Open-Source (**NOT** all is public!)

ParScale is an open-source particle-scale simulation tool

- **Continuum and shrinking-core** type of models
- **Grain models** (e.g., model oxide layer formation)

CPPPO is a data analysis tool for "heavy jobs"

- **Spatial averaging** in parallel
- Sampling & binning of raw data for on-the-fly data compression

Outlook

Announcement

OpenFOAM® User Meeting (PFAU X)

The Institute for Process and Particle Technology is pleased to host the “PFAU X” (*Palaver OpenFOAM Users Austria - the Austrian User Group*).

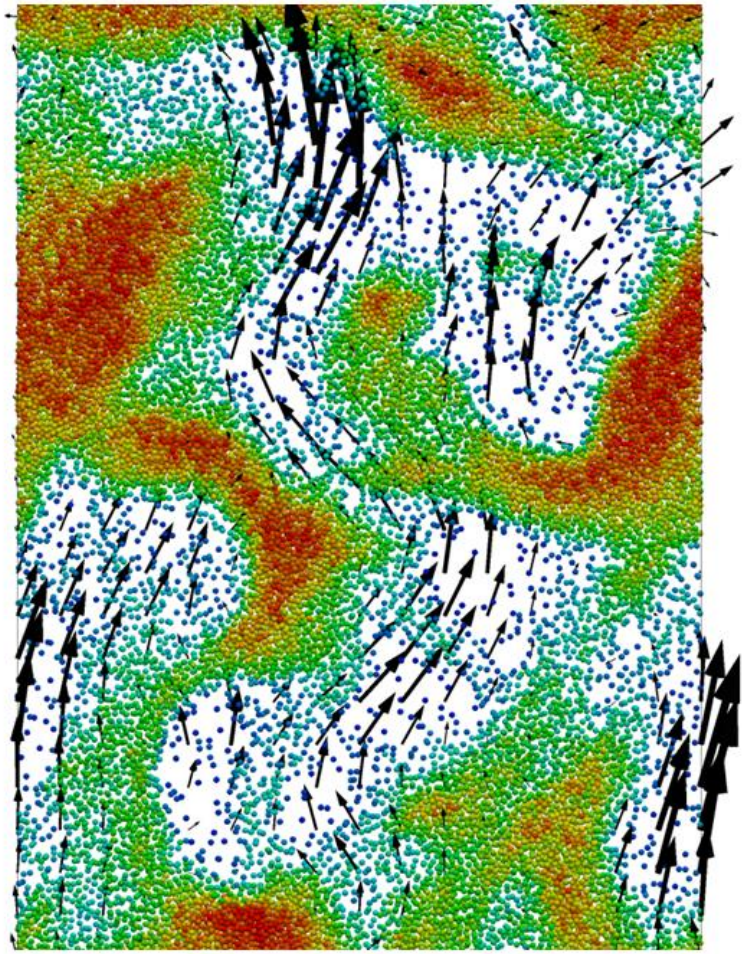
The meeting will take place on

Wednesday, July 8, 2015
10.00 a.m. - 4.00 p.m.

in

Lecture Room “HS i8”
Inffeldgasse 13, TU Graz

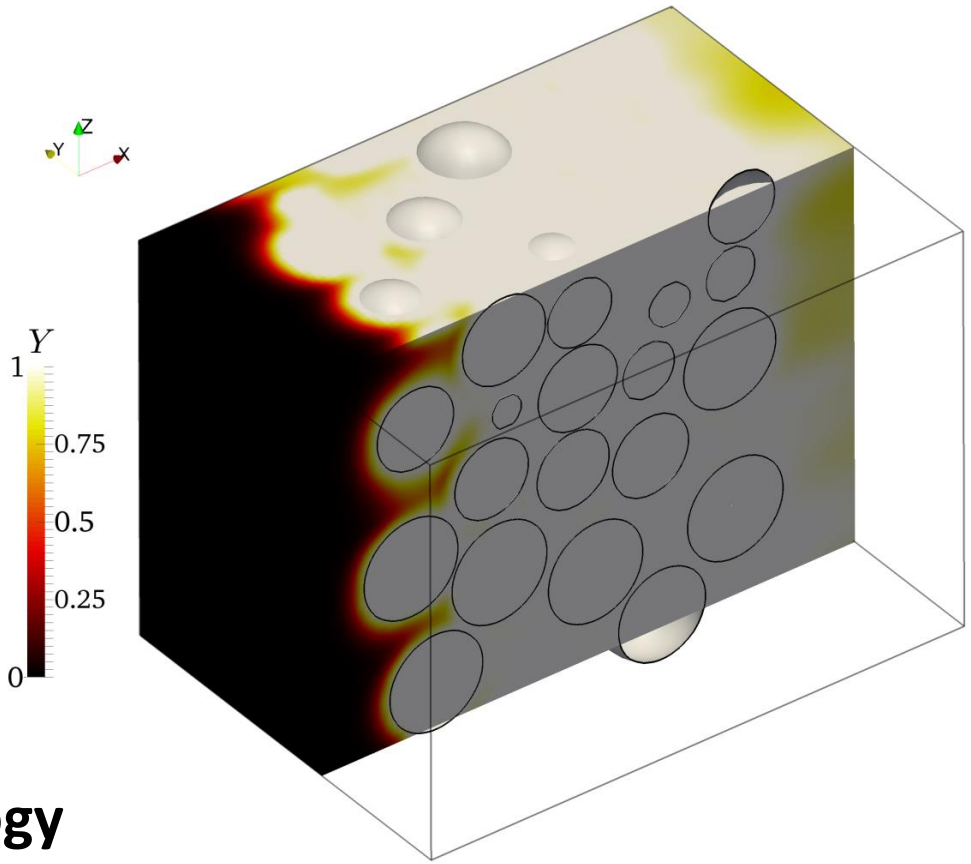
PFAU meetings are informal, and provide a channel for open discussions on finished, ongoing or future projects related to the development and usage of OpenFOAM®.



Perspectives for

Modeling Reactive Particulate Systems using Open-Source Tools

Stefan Radl,
Federico Municchi,
and Thomas Forgber



Graz University of Technology

Disclaimer

LIGGGHTS® is a registered trade mark of DCS Computing GmbH, the producer of the LIGGGHTS® software. CFDEM® is a registered trade mark of DCS Computing GmbH, the producer of the CFDEM®coupling software. This offering is not approved or endorsed by DCS Computing GmbH, the producer of the LIGGGHTS® and CFDEM® coupling software and owner of the LIGGGHTS and CFDEM® trade marks.

OpenFOAM® is a registered trade mark of OpenCFD Limited, the producer of the OpenFOAM software. This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks.

AVL Fire® is is a registered trade mark of AVL Corporation.

swak4Foam is maintained by Bernhard Gschaider