On FEM techniques for multiphase flow

Recent developments regarding Numerics and CFD Software

Stefan Turek, Otto Mierka, Raphael Münster
Institut für Angewandte Mathematik, LS III
Technische Universität Dortmund
ture@featflow.de

http://www.mathematik.tu-dortmund.de/LS3
http://www.featflow.de

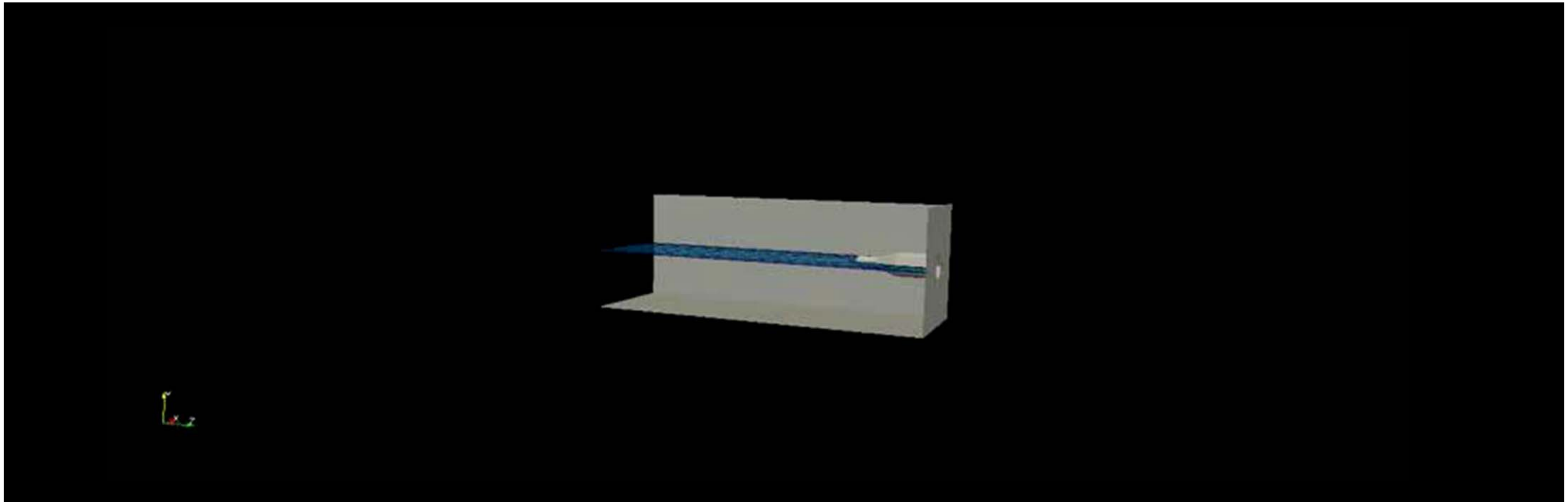
Overview & Motivation:

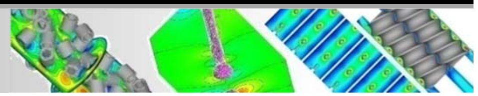
Accurate, robust, flexible and efficient simulation of multiphase problems with dynamic interfaces and complex geometries, particularly in 3D, is still a challenge!

- Mathematical Modelling
- Numerics / CFD Techniques
- Validation / Benchmarking
- HPC Techniques / Software

Vision: *Highly efficient, flexible and accurate „real life“ simulation tools based on modern Numerics and algorithms while exploiting modern hardware!*

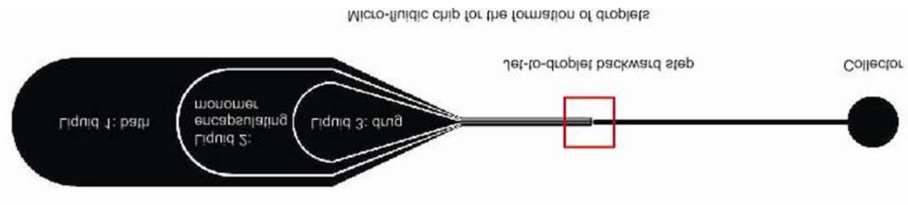
Realization: **FEATFLOW**

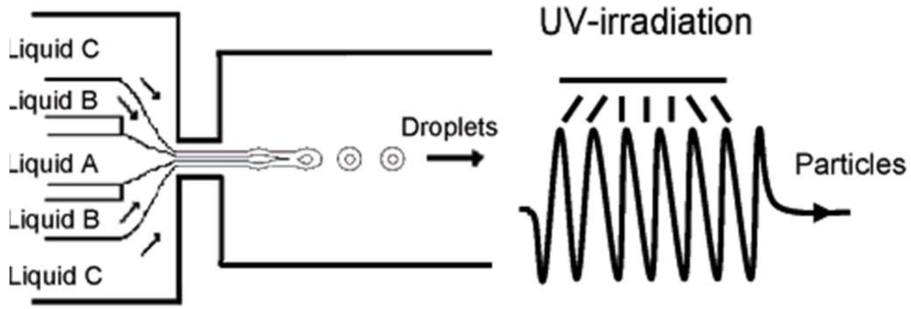


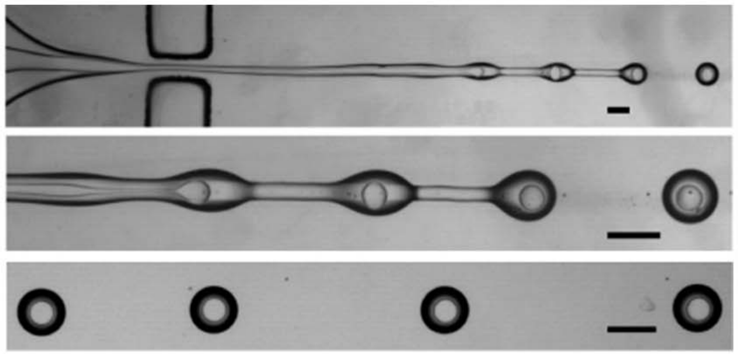


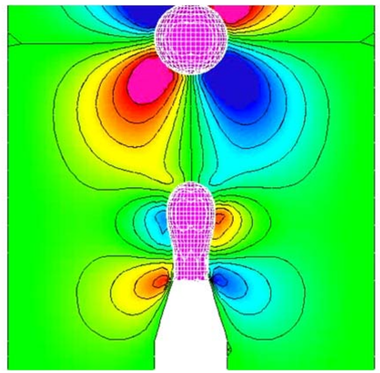
Motivation: Target Application I

- Numerical simulation of *micro-fluidic drug encapsulation (“monodisperse compound droplets”)* for application in lab-on-chip and bio-medical devices
- Polymeric “bio-degradable” outer fluid with viscoelastic effects
- Optimization of chip design w.r.t. boundary conditions, flow rates, droplet size, geometry



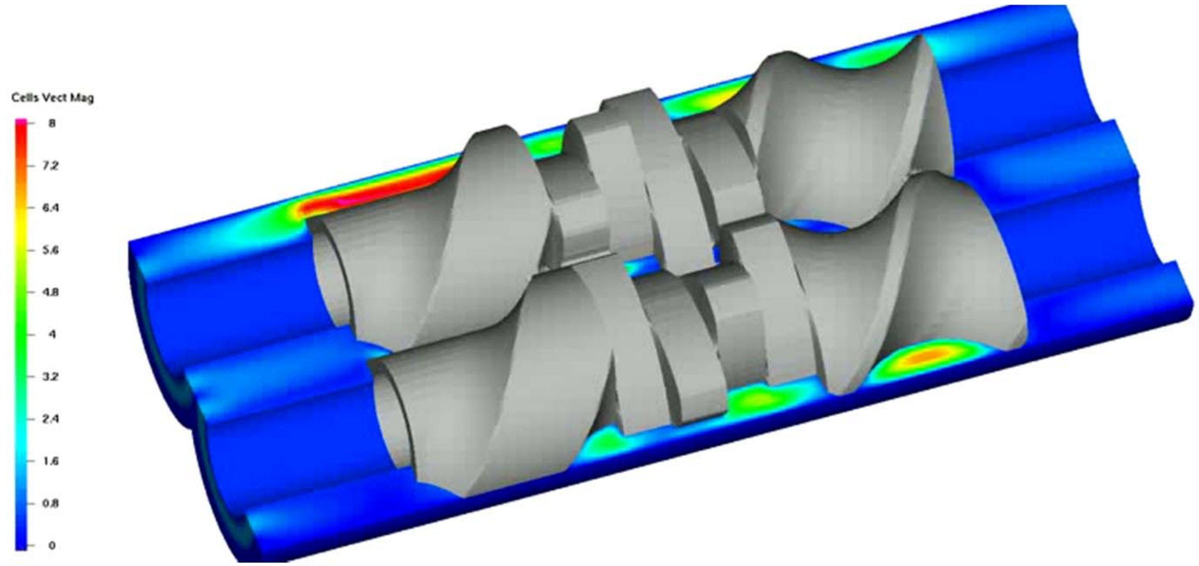


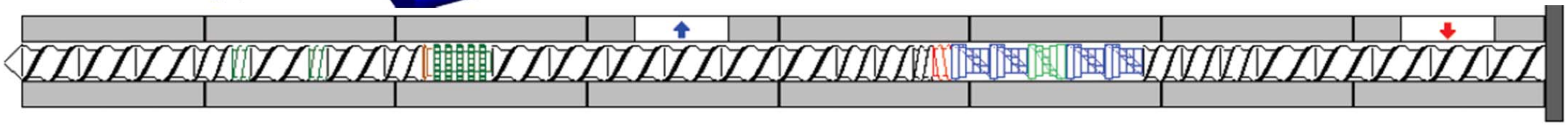




Motivation: Target Application II

- Numerical simulation of *twinscrew extruders*
- *Non-Newtonian rheological* models (shear & temperature dependent) with *non-isothermal* conditions (cooling from outside, heat production)
- Analysis of the influence of local characteristics on the global product quality, prediction of hotspots and maximum shear rates
- Optimization of torque acting on the screws, energy consumption



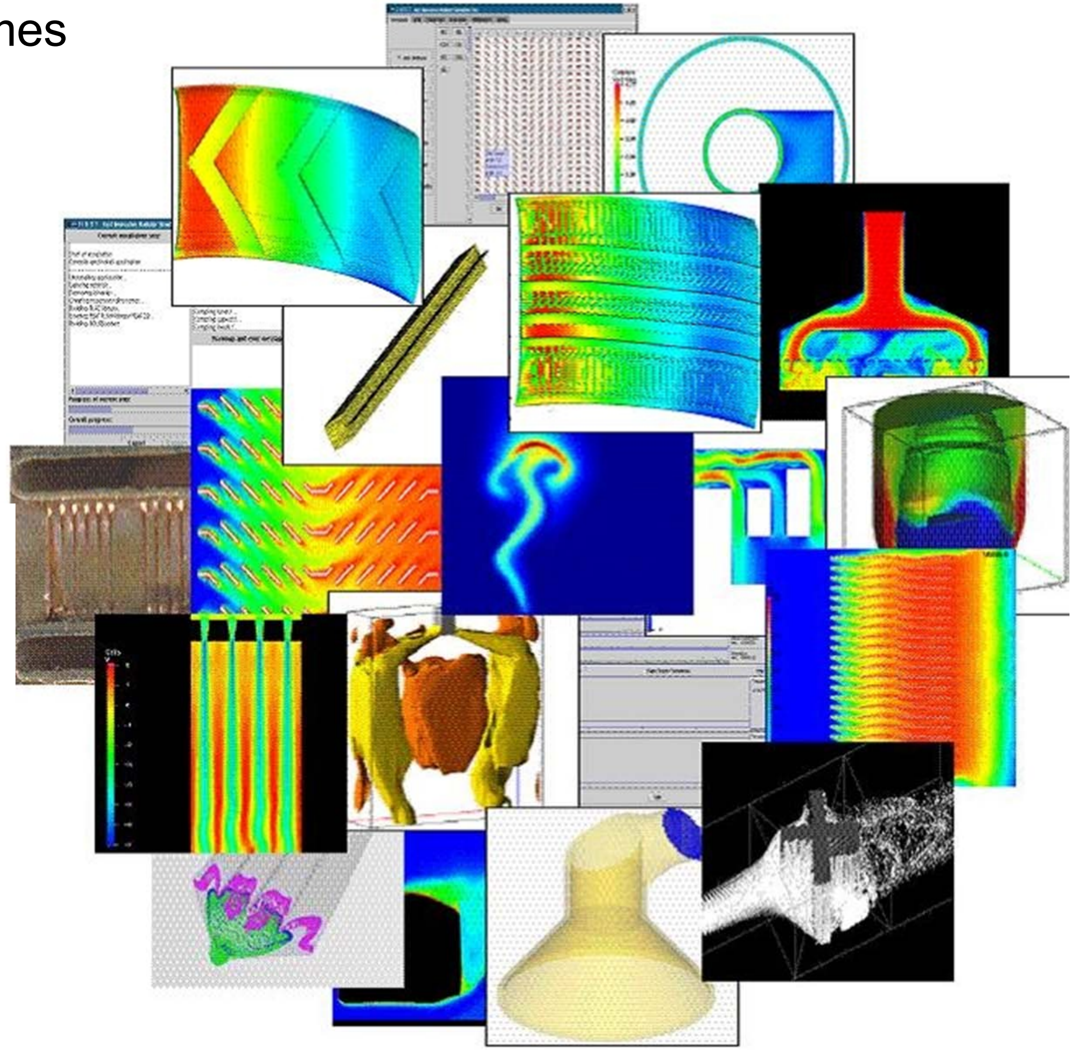


Both applications require efficient basic flow solvers
and techniques for liquid-liquid & liquid-solid interfaces
in complex (time-dependent) domains

Basic Flow Solver: FeatFlow

<u>Numerical features:</u>

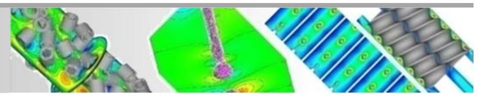
- High order FEM discretization schemes
- FCT & EO stabilization techniques
- Use of unstructured meshes
- Fictitious Boundary (FBM) methods
- Adaptive grid deformation
- Newton-Multigrid solvers



<u>HPC features:</u>

- Massive parallel
- GPU computing
- Open source

Hardware-oriented Numerics



Two phase flow (l-l) with resolved interfaces

The incompressible Navier Stokes equations

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) - \nabla\cdot\left(\mu\left[\nabla\mathbf{v} + (\nabla\mathbf{v})^{\mathrm{T}}\right]\right) + \nabla p = \mathbf{f}_{\mathrm{ST}} + \rho\mathbf{g}$$

$$\nabla\cdot\mathbf{v} = 0$$

Interface tension force

$$\mathbf{f}_{ST} = \sigma\kappa\mathbf{n}, \quad \kappa = -\nabla\cdot\mathbf{n} \quad \text{on} \quad \Gamma$$

unknown interface location

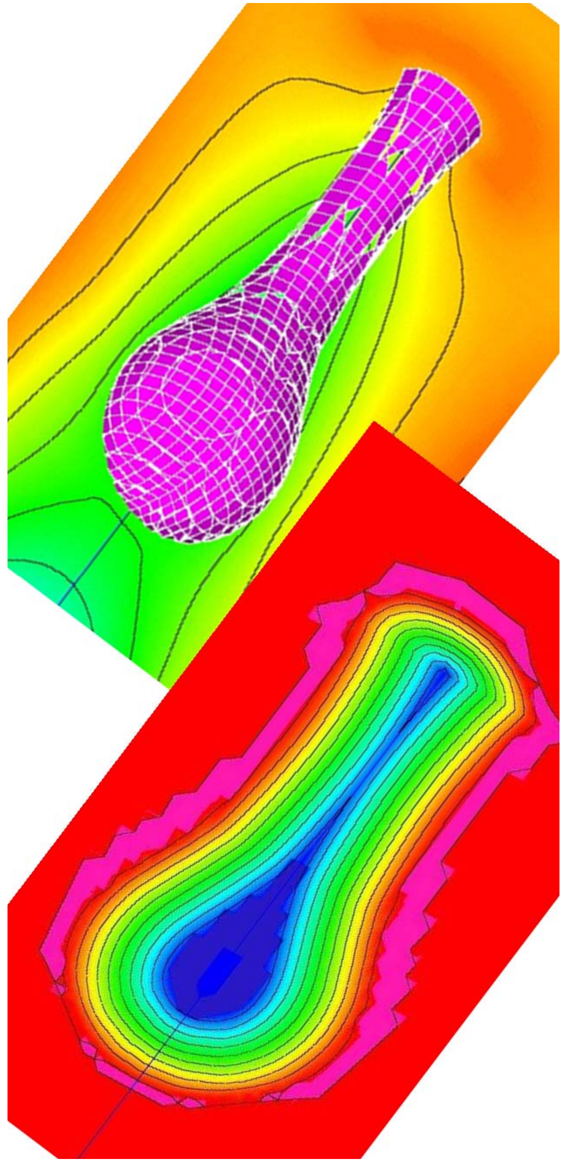
Dependency of physical quantities

$$\mu = \mu(D(\mathbf{v}),\Gamma), \quad \rho = \rho(\Gamma)$$

Interface capturing realized by **Level Set method**

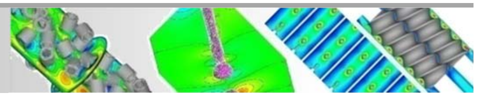
$$\frac{\partial \phi}{\partial t} + \mathbf{v}\cdot\nabla\phi = 0$$

- Exact representation of the interface
- Natural treatment of topological changes
- Provides derived geometrical quantities (**n,** κ)



Two phase flow (l-l) with resolved interfaces

Problems and Challenges
• **Steep gradients** of the velocity field and of other physical quantities near the interface (oscillations!)
• **Reinitialization** w.r.t. distance field (artificial movement of the interface → mass loss, how often to perform?)
• **Mass conservation** (during advection and reinitialization of the Level Set function)
• Representation of **surface tension**: CSF, Line Integral, Laplace-Beltrami, Phasefield, etc.
• **Explicit** or **implicit** treatment (→ *Capillary Time Step* restriction?)



Two phase flow (l-l) with resolved interfaces

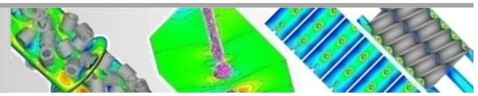
Steep changes of physical quantities:

1) Elementwise averaging of the physical properties (prevents oscillations):

   $\rho_e = x\rho_1 + (1-x)\rho_2, \quad \mu_e = x\mu_1 + (1-x)\mu_2$ — *x* is the volume fraction

2) Flow part: Extension of nonlinear stabilization schemes (FCT, TVD, EO-FEM) for the momentum equation for LBB stable element pairs with discontinuous pressure (Q2/P1)

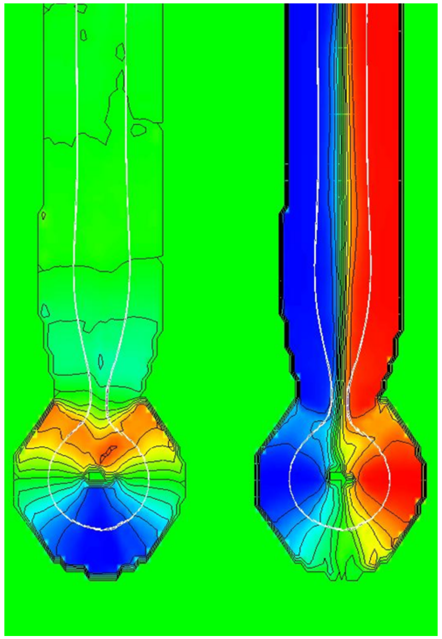
3) Interface tracking part with DG(1)-FEM: Flux limiters satisfying LED requirements



Two phase flow (l-l) with resolved interfaces

Reinitialization
- Mainly required in the vicinity of the interface
- How often to perform?
- Which realization to implement?
- Efficient parallelization (3D!)

Alternatives
- Brute force (introducing new points at the zero level set)
- Fast sweeping („advancing front“ upwind type formulas)
- Fast marching
- Algebraic Newton method
- **Hyperbolic PDE approach**
- many more…..



Globally defined normal vectors

Maintaining the signed distance function by PDE reinitialization

$$\frac{\partial \phi}{\partial \tau} + \mathbf{u} \cdot \nabla \phi = S(\phi) \qquad \mathbf{u} = S(\phi)\frac{\nabla \phi}{|\nabla \phi|} \quad \Leftrightarrow \quad |\nabla \phi| = 1$$

Problems:
- What to do with the sign function at the interface? (smoothing?)
- How to handle the underlying non-linearity?
- How often to perform? (expensive → steady state)

Two phase flow (l-l) with resolved interfaces

Fine-tuned reinitialization

<u>Our reinitialization is performed in combination of 2 ingredients:</u>

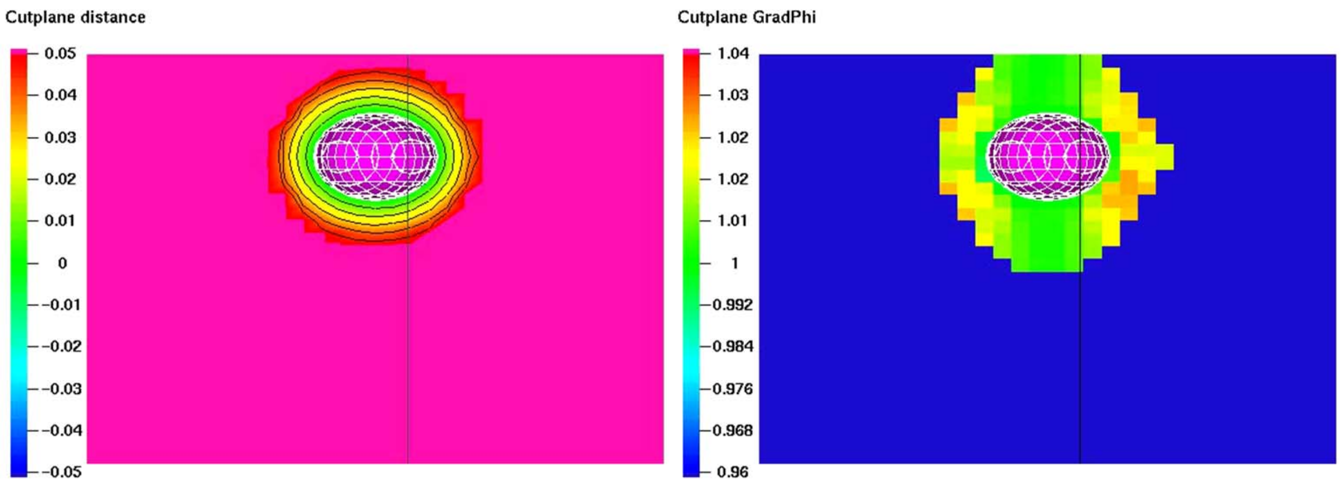
1) Elements intersected by the interface are modified w.r.t. the slope of the distance distribution („Parolini trick“) such that

$$|\nabla \phi| = 1$$

2) Far field reinitialization: realization is based on the PDE approach („FBM“), but it does not require smoothening of the distance function!

In addition: continuous projection of the interface (smoothening of the discontinuous P_1 based distance function)

$$\phi_{P_1} \xrightarrow{L_2 \text{ projection}} \phi_{Q_1} \xrightarrow{L_2 \text{ projection}} \phi_{P_1}$$



Two phase flow (l-l) with resolved interfaces

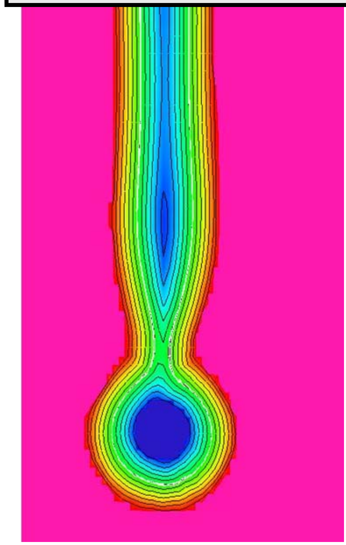
Continuum
Surface
Force

- Transformation of the surface integrals to volume integrals with the help of a <u>regularized</u> Dirac delta function δ
- Requires globally defined normals and curvature
- Reduction of spurious oscillations

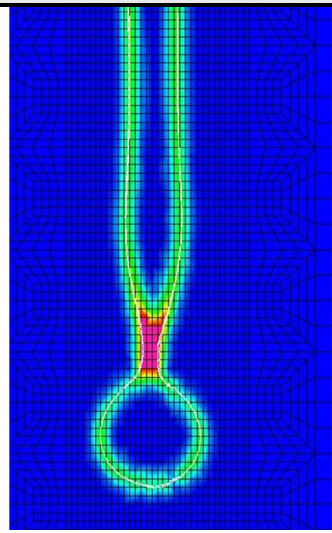
$$\mathbf{f}_{\mathrm{ST}} = \sigma\kappa\mathbf{n}\delta(x,\varepsilon)$$

$$\mathbf{n}_{P_1} \xrightarrow{\mathrm{L}_2\text{ projection}} \mathbf{n}_{Q_1} \qquad \text{continuous normal field}$$

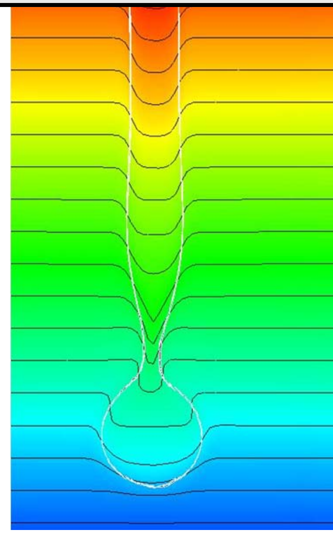
$$\kappa_{Q_1} = \frac{\int_\Omega \nabla\cdot\mathbf{n}_{Q_1}\, d\mathbf{x}}{\int_\Omega d\mathbf{x}} \qquad \text{continuous curvature field}$$



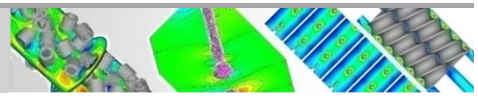
Level Set distribution



Distribution of the smoothed surface tension force $(\sigma\kappa\delta)_{Q_1}$



Resulting pressure distribution



Two phase flow (l-l) with resolved interfaces

<u>Surface Tension</u>: Semi-implicit CSF formulation based on Laplace-Beltrami

$$\mathbf{f}_{\mathrm{ST}} = \int_\Omega \sigma\kappa\hat{\mathbf{n}}\cdot\mathbf{v}\,\delta(\Gamma,\mathbf{x})\,d\mathbf{x} \qquad = \int_\Omega \sigma\left(\underline{\Delta}\mathbf{x}|_\Gamma\right)\cdot\left(\mathbf{v}\,\delta(\Gamma,\mathbf{x})\right)d\mathbf{x}$$

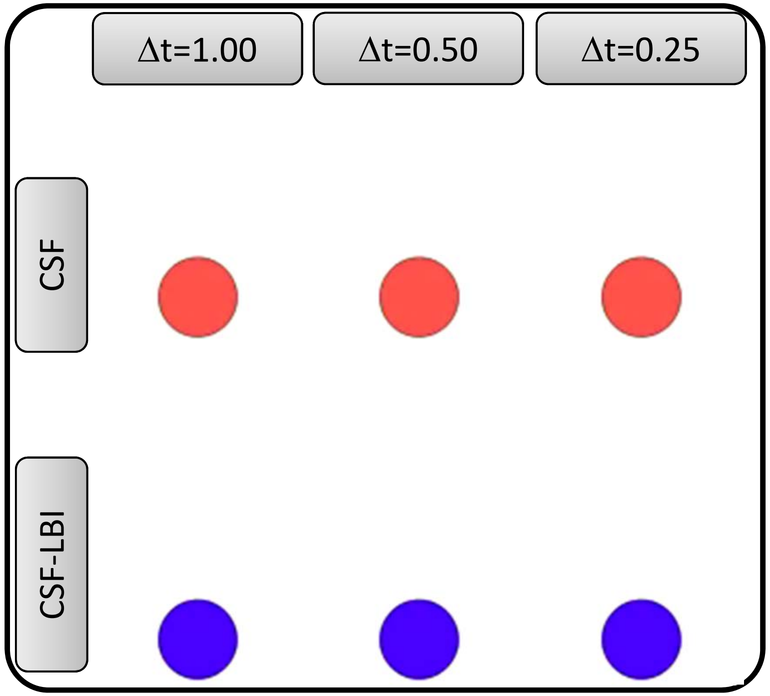
$$= -\int_\Omega \sigma\underline{\nabla}\mathbf{x}|_\Gamma\cdot\underline{\nabla}\left(\mathbf{v}\,\delta(\Gamma,\mathbf{x})\right)d\mathbf{x} = -\int_\Omega \sigma\underline{\nabla}\mathbf{x}|_\Gamma\cdot\underline{\nabla}\mathbf{v}\,\delta(\Gamma,\mathbf{x})\,d\mathbf{x}$$

Application of the semi-implicit time integration yields $\mathbf{x}|_{\Gamma^{n+1}} = \mathbf{x}|_{\Gamma^n} + \Delta t\,\mathbf{u}^{n+1}$

$$\Downarrow$$

$$\mathbf{f}_{\mathrm{ST}} = -\int_\Omega \sigma\,\delta_\varepsilon\left(dist(\Gamma^n,\mathbf{x})\right)\underline{\nabla}\tilde{\mathbf{x}}|_\Gamma^n\cdot\underline{\nabla}\mathbf{v}\,d\mathbf{x}$$

$$-\Delta t^{n+1}\int_\Omega \sigma\,\delta_\varepsilon\left(dist(\Gamma^n,\mathbf{x})\right)\underline{\nabla}\mathbf{u}^{n+1}\cdot\underline{\nabla}\mathbf{v}\,d\mathbf{x}$$



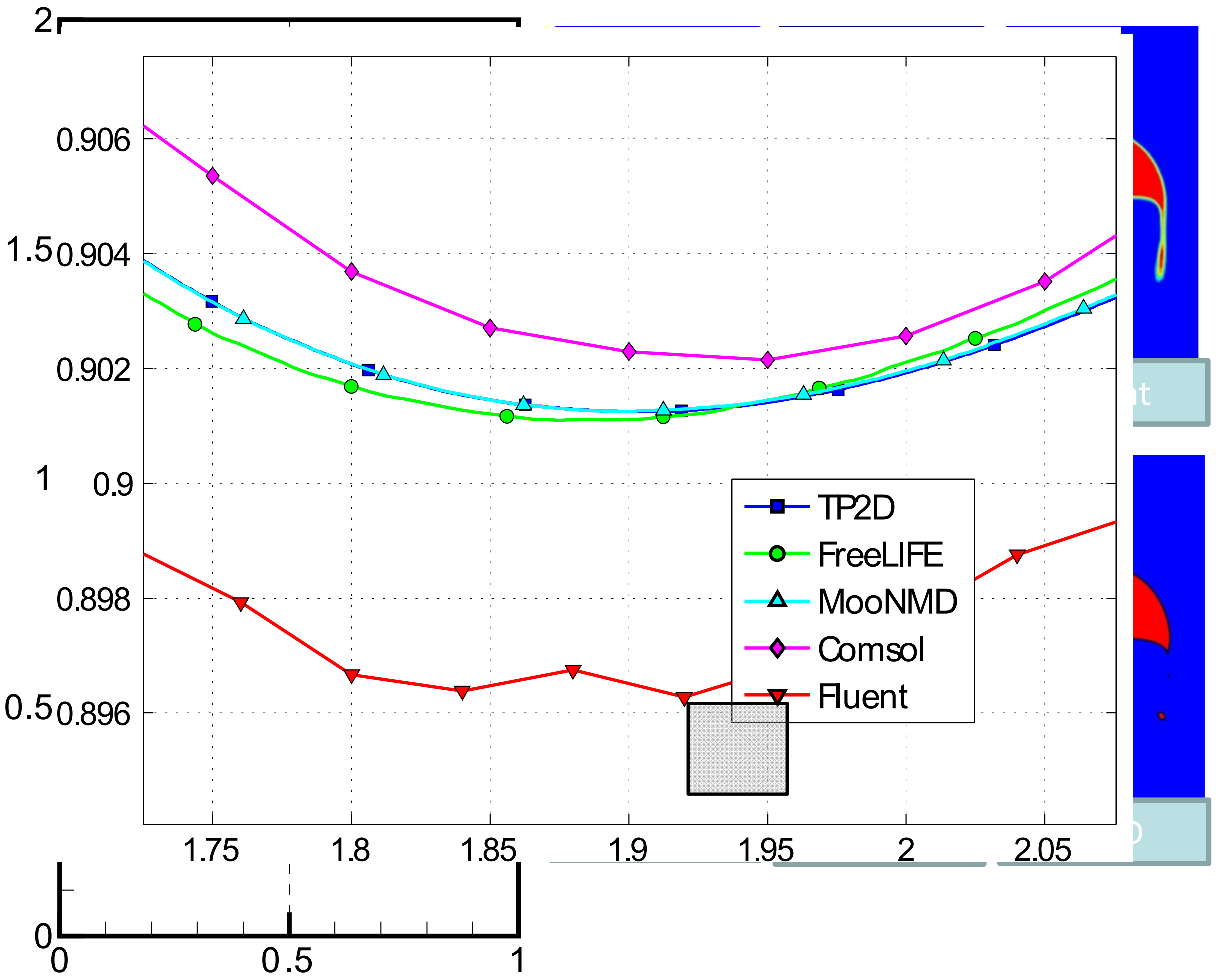
Advantages

- Relaxes *Capillary Time Step* restriction
- „Optimal“ for FEM-Level Set approach due to global information

Benchmarking

2D Bubble Benchmarks

http://www.featflow.de/beta/en/benchmarks/



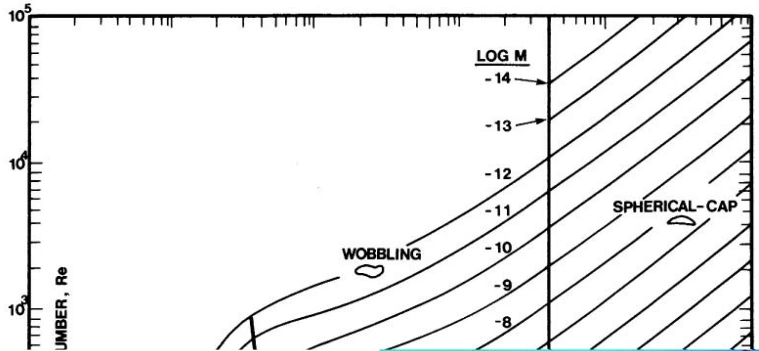
Benchmark quantities

Center of mass	$\mathbf{x}_c = \dfrac{\int_{\Omega_2} \mathbf{x}\, dx}{\int_{\Omega_2} 1\, dx}$
Mean rise velocity	$\mathbf{U}_c = \dfrac{\int_{\Omega_2} \mathbf{u}\, dx}{\int_{\Omega_2} 1\, dx}$
Circularity	$¢ = \dfrac{P_a}{P_b} = \dfrac{\pi d_a}{P_b}$

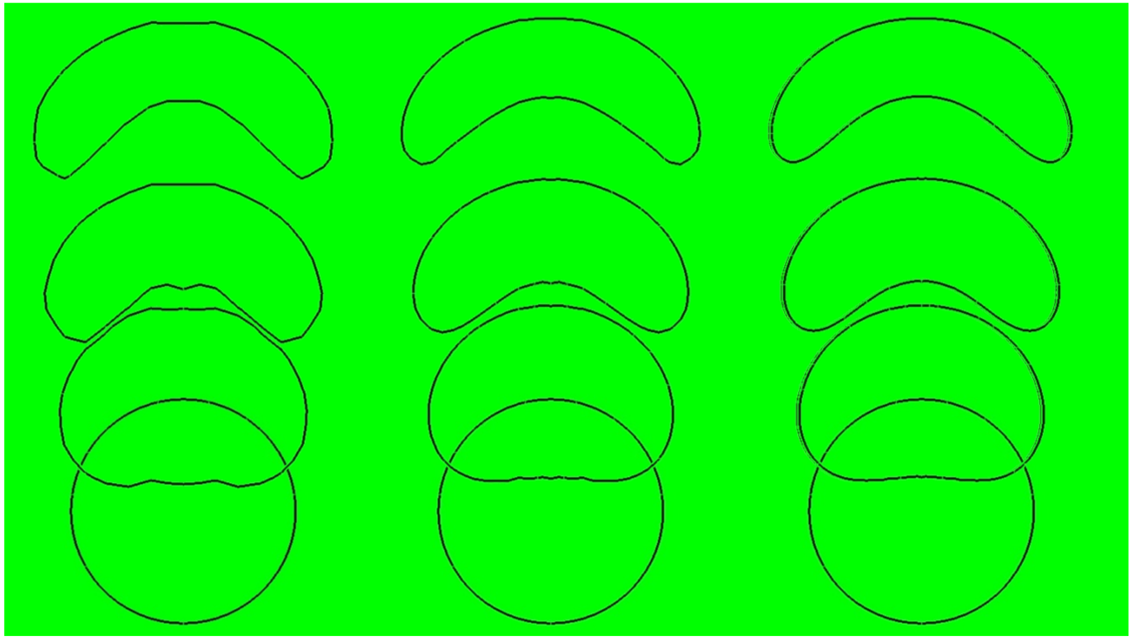
Hysing, S.; Turek, S.; Kuzmin, D.; Parolini, N.; Burman, E.; Ganesan, S.; Tobiska, L.: Quantitative benchmark computations of two-dimensional bubble dynamics, International Journal for Numerical Methods in Fluids, 2009

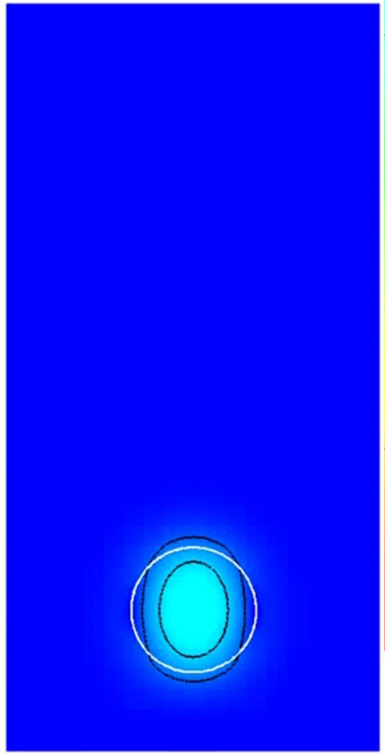
3D convergence analysis for large density jumps

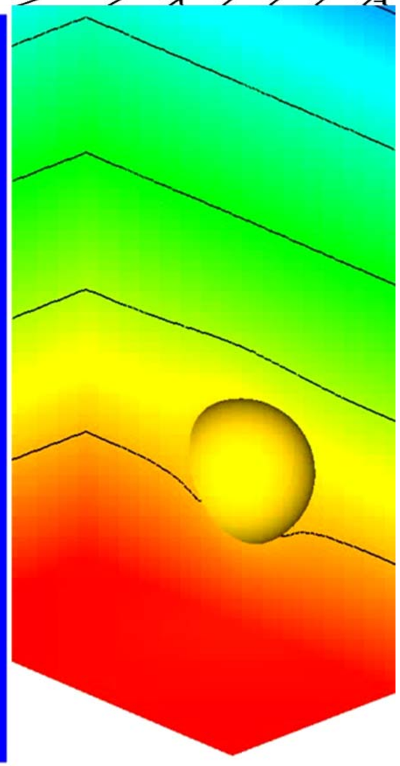
Rising bubble problem for Eo = 60, Re = 34
Density jump 1:100



Level 2 Level 3 Level 4







Benchmarking with experimental results

Continuous phase:

Glucose-Water mixture

$\mu_D = 500\, mPa\, s$

$\rho_D = 972\, kg\, m^{-3}$

$\dot{V}_D = 3{,}64\, ml\, min^{-1}$

$\sigma_{CD} = 0{,}034\, N\, m^{-1}$

Silicon oil

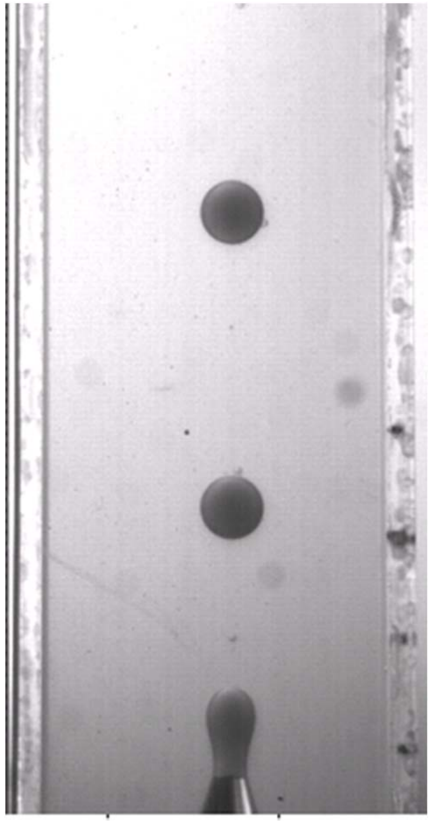
$\mu_C = 500\, mPa\, s$

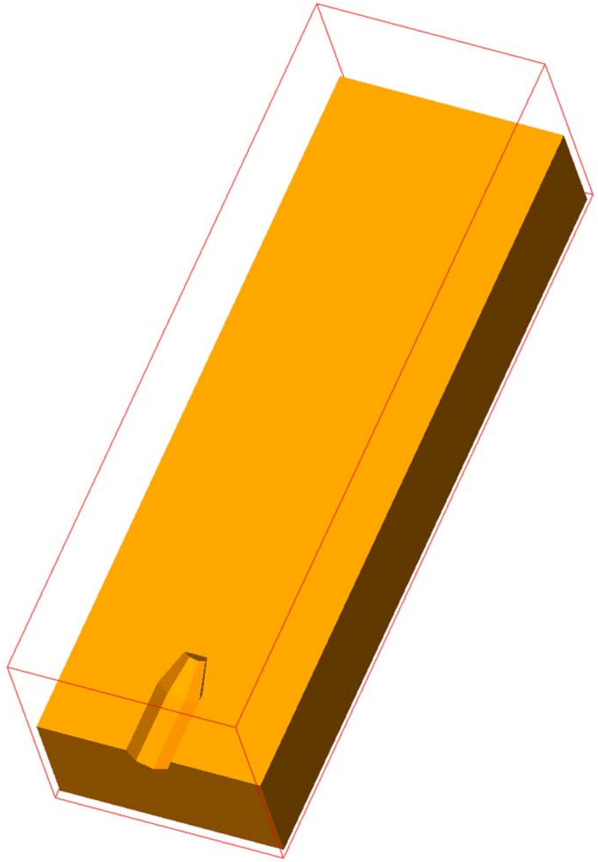
$\rho_C = 1340\, kg\, m^{-3}$

$\dot{V}_C = 99{,}04\, ml min^{-1}$

Dispersed phase:

Experimental setup with **AG Walzel** **(BCI/Dortmund)**

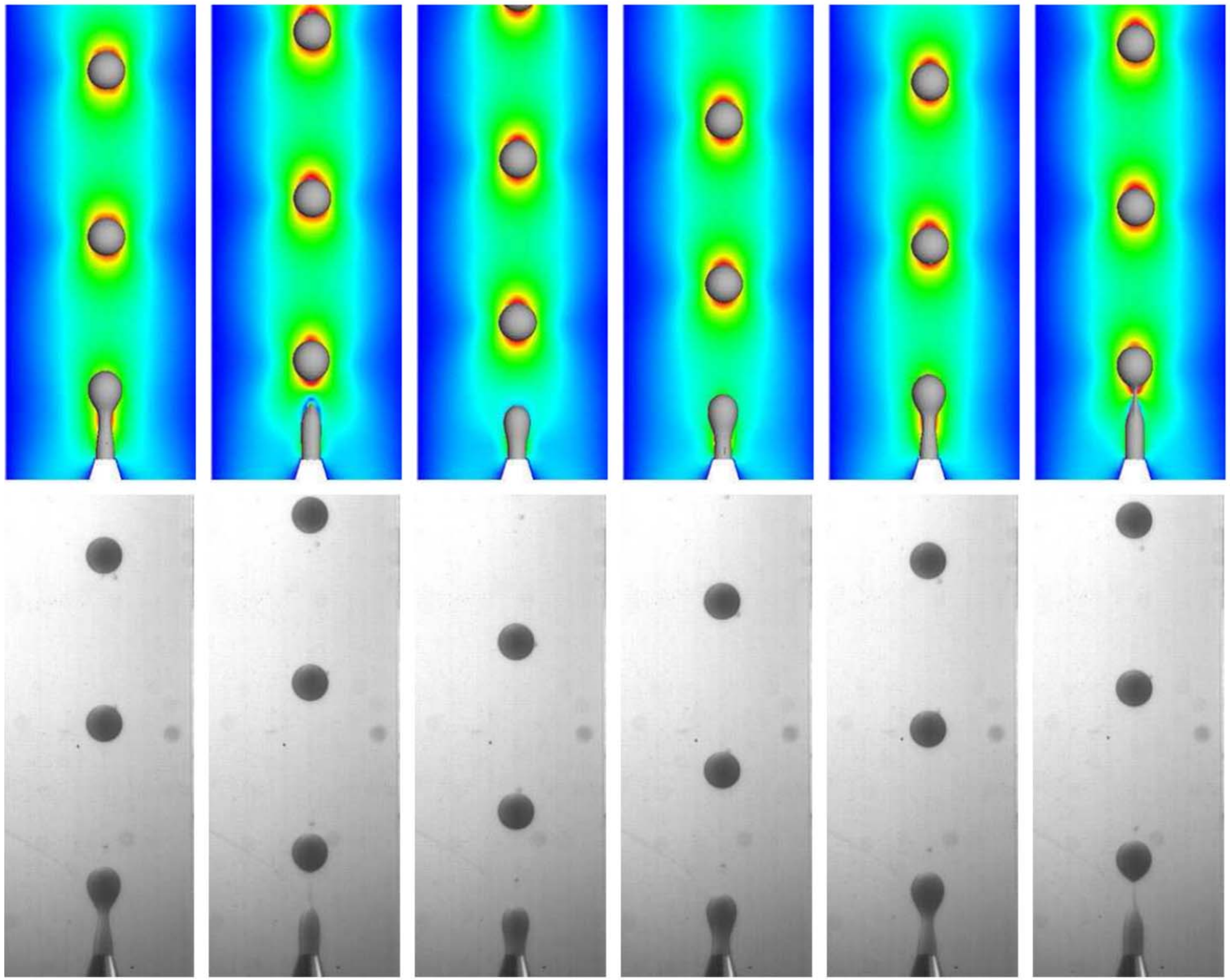


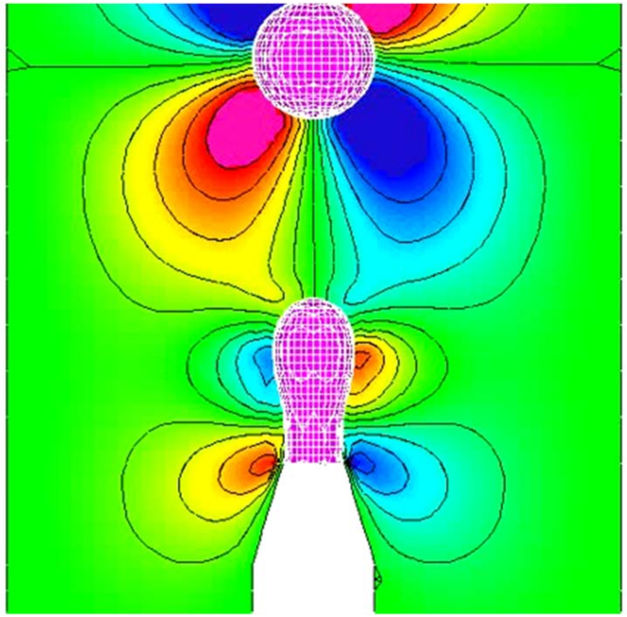


Validation parameters:

- frequency of droplet generation
- droplet size
- stream length

Benchmarking with experimental results





	Separation frequency [Hz]	Drople t size [dm]	Stream Length [dm]
Exp	0,58	0,062	0,122
Sim	0,6	0,058	0,102

Tailored monodisperse droplets via modulation

In case of monodisperse droplet generation:

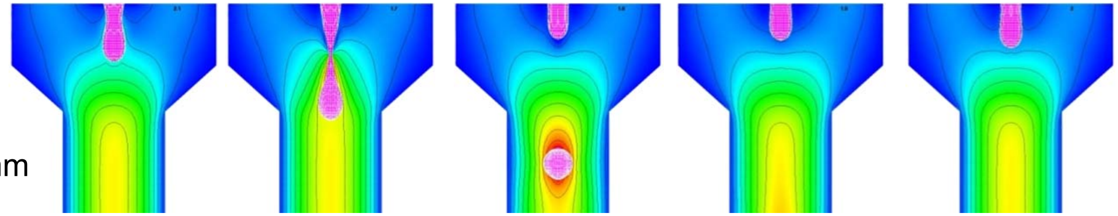
$$\dot{V}_D = f V_{\text{droplet}}$$

Influencable variables

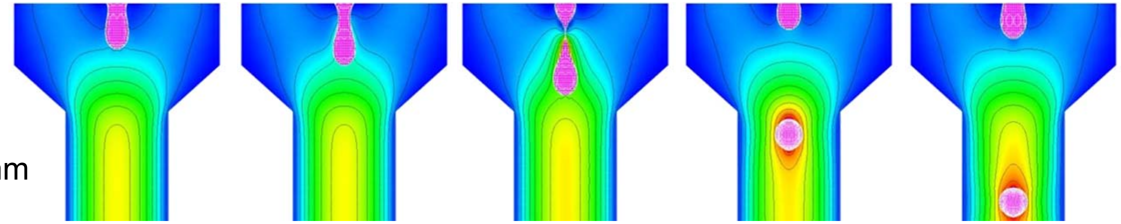
On the level of the process:
- Flowrates
- Modulation frequency
- Modulation amplitude

Geometrical changes:
- Capillary size
- Contraction angle
- Contraction ratio

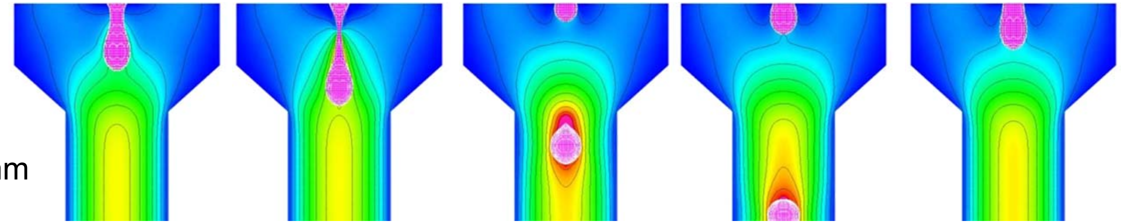
No Regulation
Flowrate: 100%
Capillary: STD
Droplet size: 5.2mm



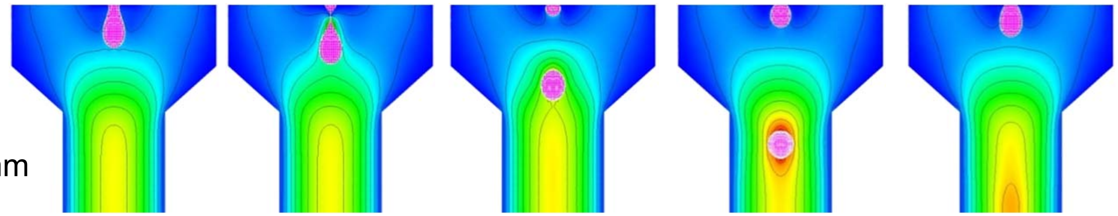
Regulated
Flowrate: 100%
Capillary: STD
Droplet size: 5.0mm



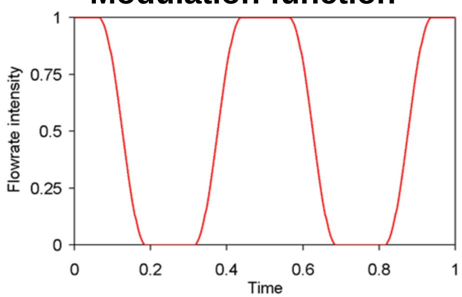
Regulated
Flowrate: 150%
Capillary: STD
Droplet size: 5.7mm

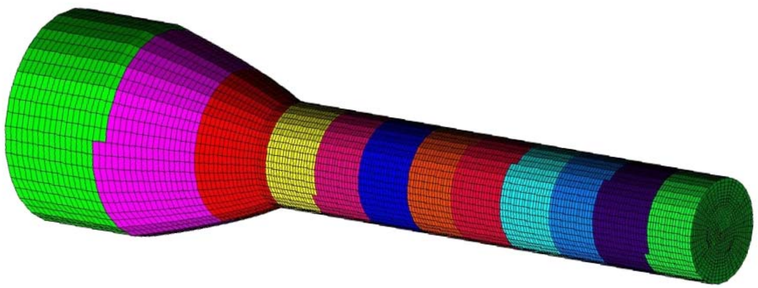


Regulated
Flowrate: 75%
Capillary: STD
Droplet size: 4.5mm



Modulation function

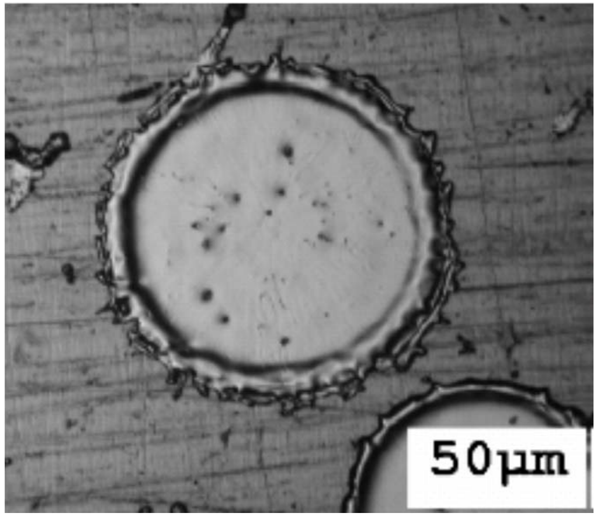


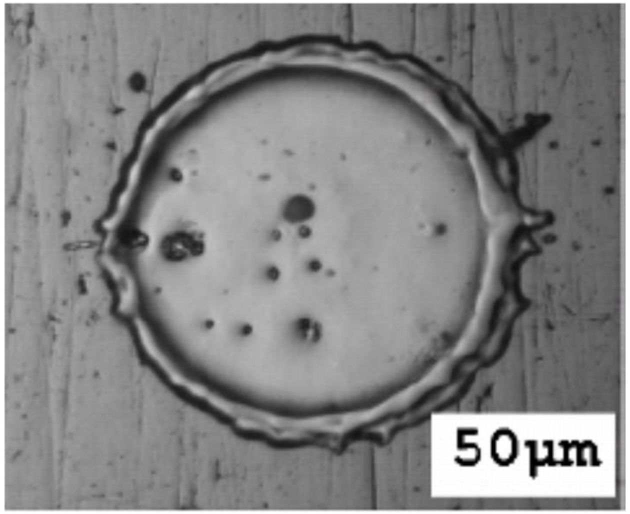


Resulting operation envelope:
- Size: 4.5 mm – 5.7 mm
- Volume: 0.38 cm^3 – 0.77 cm^3

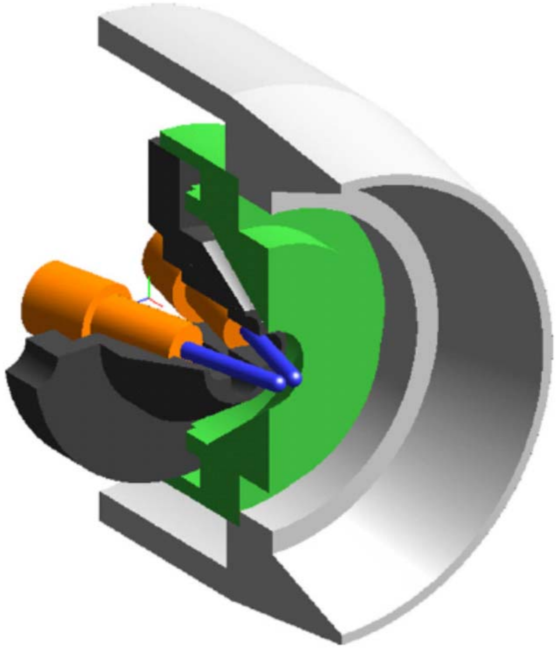
Next step: Interaction of droplets with surfaces

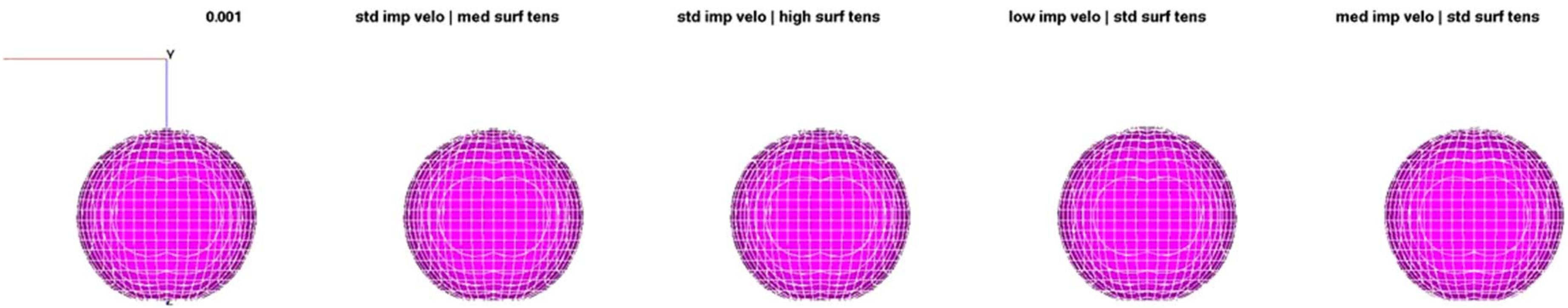
Pourmousa 2007



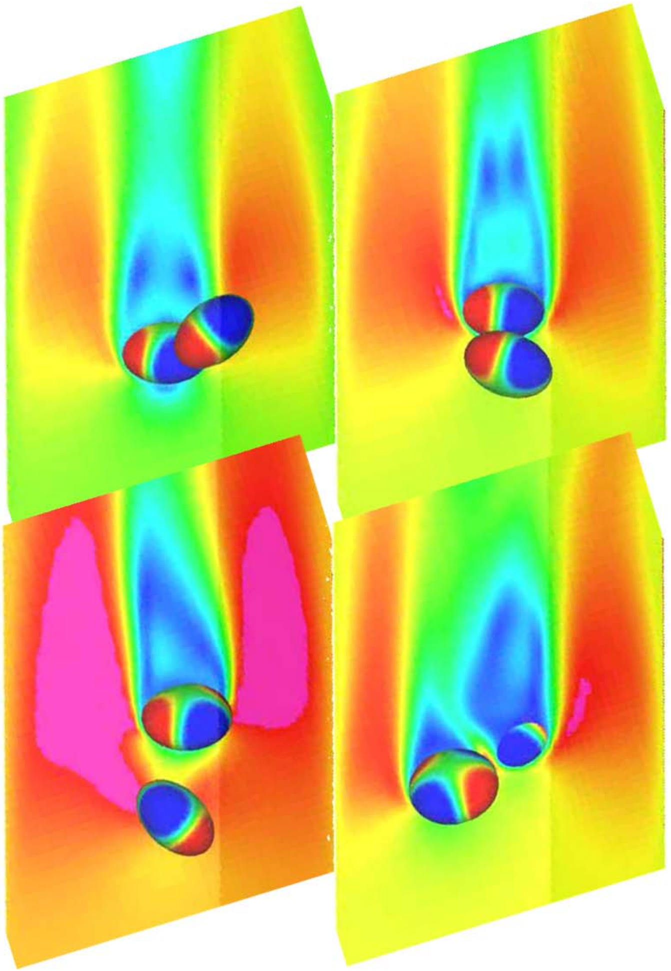


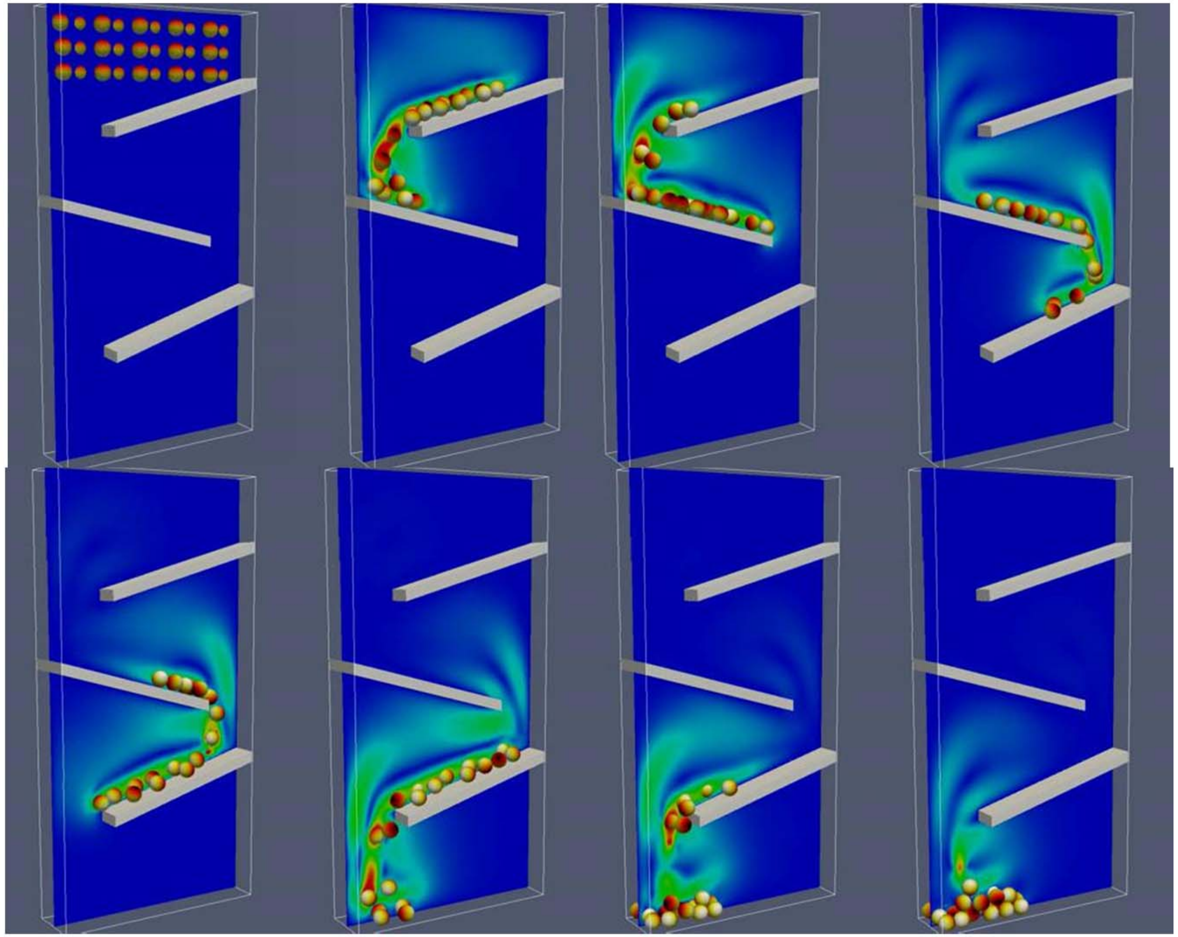
Bolot et al. 2008





Now: Particulate Flow with Solid-Liquid Interfaces





Two phase flow (s-l) with resolved interfaces

- **Fluid** motion is governed by the **Navier-Stokes equations**
- **Particle** motion is described by **Newton-Euler equations**

$$M_p \frac{dU_p}{dt} = F_p + F_{ex,col} + (\Delta M_p) g, \qquad I_p \frac{d\omega_p}{dt} = T_p - \omega_p \times (I_p \omega_p)$$

Hydrodynamic force

Torque

$$F_p = -\int_{\Gamma_p} \sigma \cdot n_p \, d\Gamma_p$$

Postprocessing the actual flow field

$$T_p = -\int_{\Gamma_p} (X - X_p) \times (\sigma \cdot n_p) \, d\Gamma_p$$

Fictitious Boundary Method

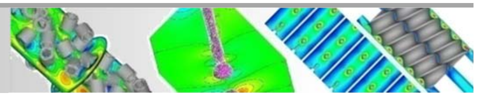
- Surface integral is replaced by volume integral
- Use of monitor function (liquid/solid)

$$\alpha_p(X) = \begin{cases} 1 & \text{for} \quad X \in \Omega_p \\ 0 & \text{for} \quad X \in \Omega_f \end{cases}$$

- Normal to particle surface vector is non-zero only at the surface of particles $n_p = \nabla \alpha_p$

$$F_p = -\int_{\Gamma_p} \sigma \cdot n_p \, d\Gamma_p = -\int_{\Omega_T} \sigma \cdot \nabla \alpha_p \, d\Omega_T$$

$$T_p = -\int_{\Gamma_p} (X - X_p) \times (\sigma \cdot n_p) \, d\Gamma_p = -\int_{\Omega_T} (X - X_p) \times (\sigma \cdot \nabla \alpha_P) \, d\Omega_T$$



Two phase flow (s-l) with resolved interfaces

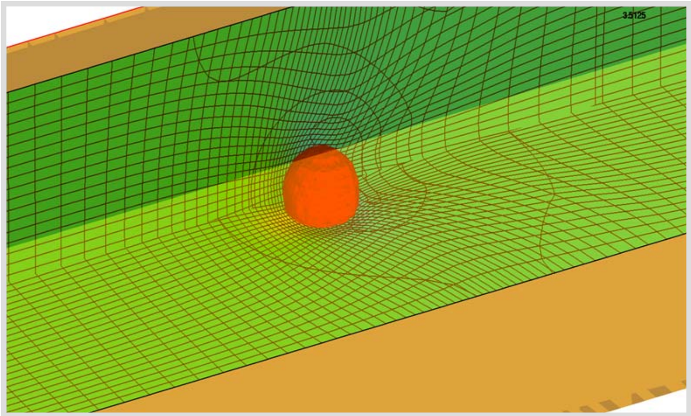
Fictitious Boundary Method

For computed $U_p^{n+1}, \omega_p^{n+1}$ → Position update: $\frac{dX_p}{dt} = U_p,$ Angle update: $\frac{d\theta_p}{dt} = \omega_p$ → $X_p^{n+1}, \theta_p^{n+1}$

Velocity "boundary condition" imposed for particles:

$$u(X) = U_p + \omega_p \times (X - X_p)$$

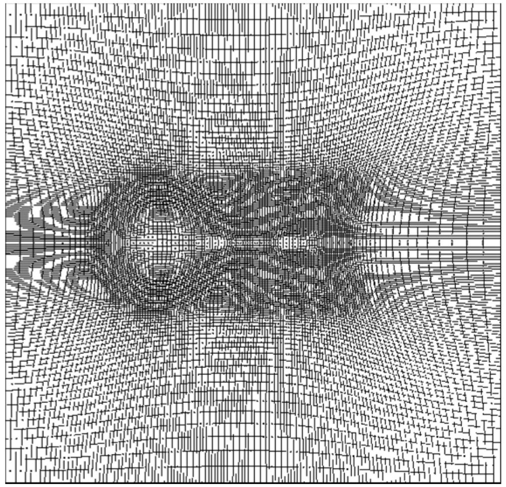
- supports HPC concepts (fixed data structures)
- easy grid generator
- relatively low resolution
  - Brute force → Finer mesh resolution
  - High resolution interpolation functions
  - **Grid deformation** (+ monitor function)



Grid Deformation Method

Idea : construct transformation $\phi,\ x = \phi(\xi, t)$ with $\det \nabla \phi = f$

⇨ local mesh area $\approx f$

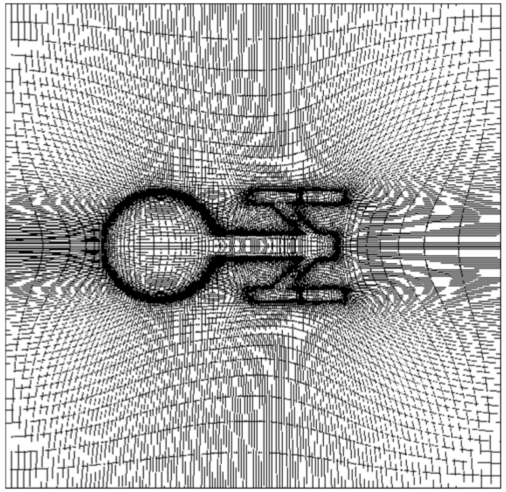


1. Compute monitor function $f(x,t) > 0,\ f \in C^1$
and

$$\int_\Omega f^{-1}(x,t)\,dx = |\Omega|, \quad \forall t \in [0,1]$$

2. Solve $(t \in [0,1])$

$$\Delta v(\xi, t) = -\frac{\partial}{\partial t}\left(\frac{1}{f(\xi,t)}\right), \quad \left.\frac{\partial v}{\partial n}\right|_{\partial\Omega} = 0$$



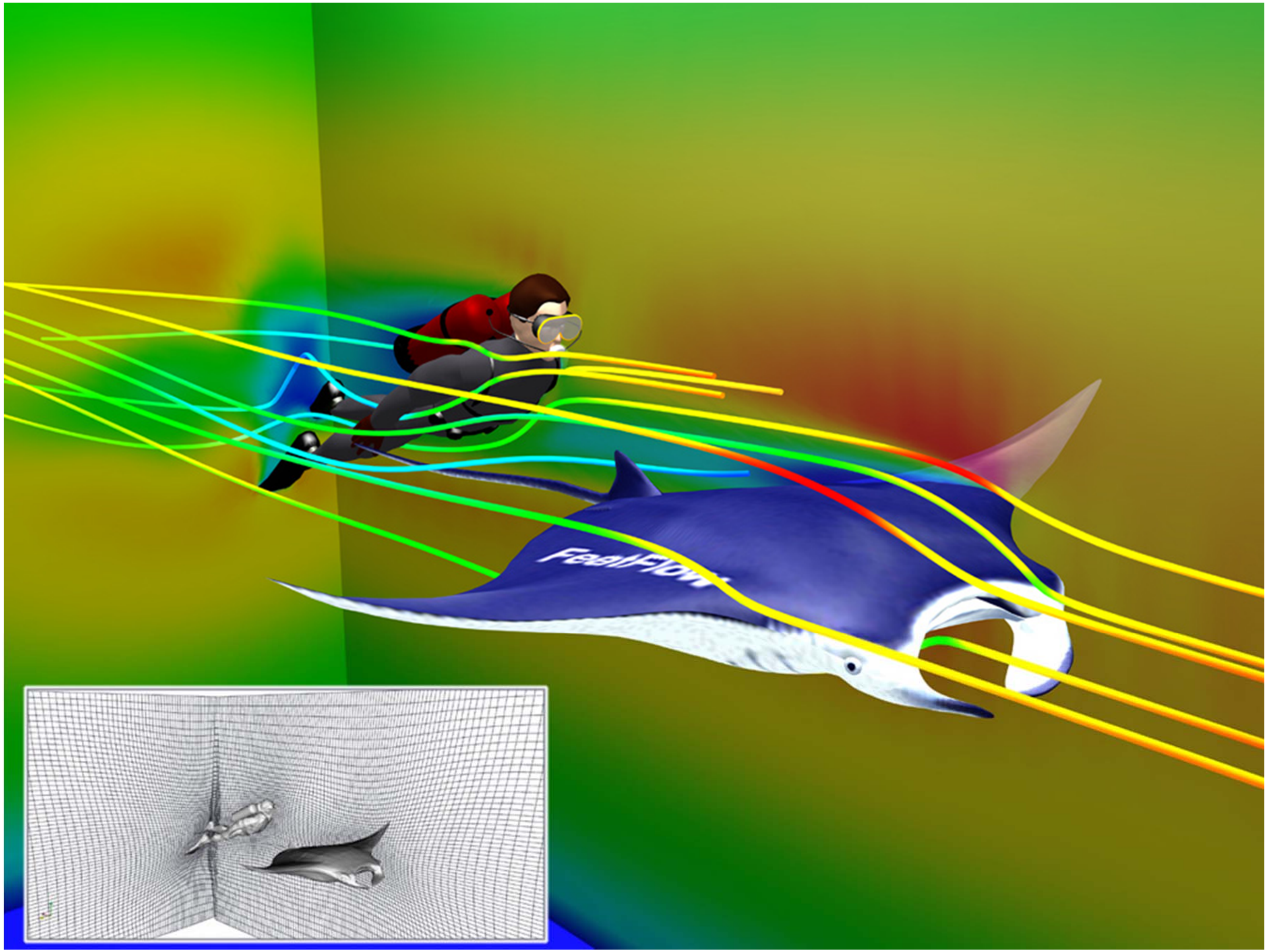
3. Solve the ODE system

$$\frac{\partial}{\partial t}\phi(\xi, t) = f(\phi(\xi,t),t)\nabla v(\phi(\xi,t),t)$$

new grid points: $x_i = \phi(\xi_i, 1)$

Grid deformation preserves the (local) logical structure of the grid

Generalized Tensorproduct Meshes

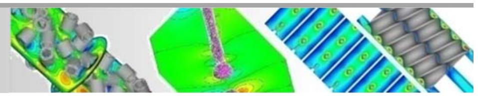


Operator-Splitting Approach

The algorithm for $t^n \rightarrow t^{n+1}$ consists of the following 5 substeps

1. Fluid velocity and pressure: $NSE\left(u_f^{n+1}, p^{n+1}\right) = BC\left(\Omega_p^n, u_p^n\right)$
2. Calculate hydrodynamic forces: F_p^{n+1}
3. Calculate velocity of particles: $u_p^{n+1} = g\left(F_p^{n+1}\right)$ (collision model)
4. Update position of particles: $\Omega_p^{n+1} = f\left(u_p^{n+1}\right)$
5. Align new mesh

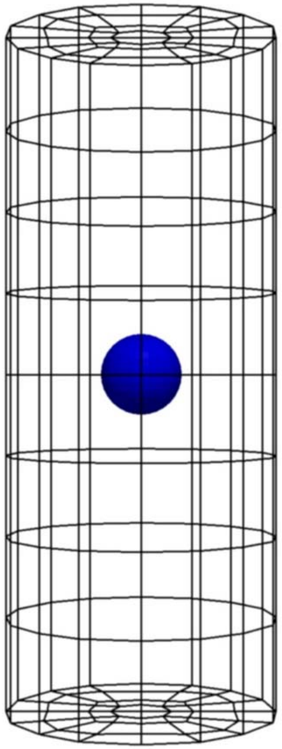
→ Required: efficient calculation of hydrodynamic forces
→ Required: efficient treatment of particle interaction (?)
→ Required: fast (nonstationary) Navier-Stokes solvers

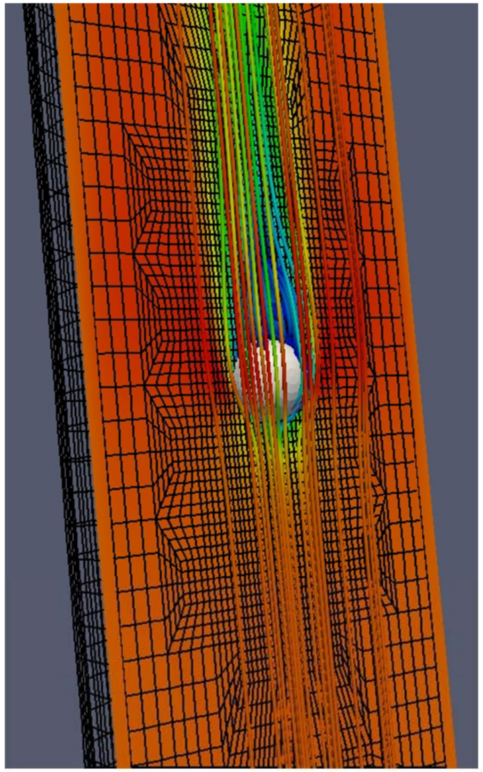


Benchmarking and Validation

Free fall of particles:

- Terminal velocity
- Different physical parameters
- Different geometrical parameters





Münster, R.; Mierka, O.; Turek, S.: Finite Element fictitious boundary methods (FEM-FBM) for 3D particulate flow, IJNMF, 2010, accepted

$d_s = 0.3, \quad \rho_s = 1.14$

ν	$U_{featflow}$	U_{exp}	Relative error (%)
0.02	5.885	6.283	6.33
0.05	4.133	3.972	4.05
0.1	2.588	2.426	6.66
0.2	1.492	1.401	6.50

$d_s = 0.2, \quad \rho_s = 1.14$

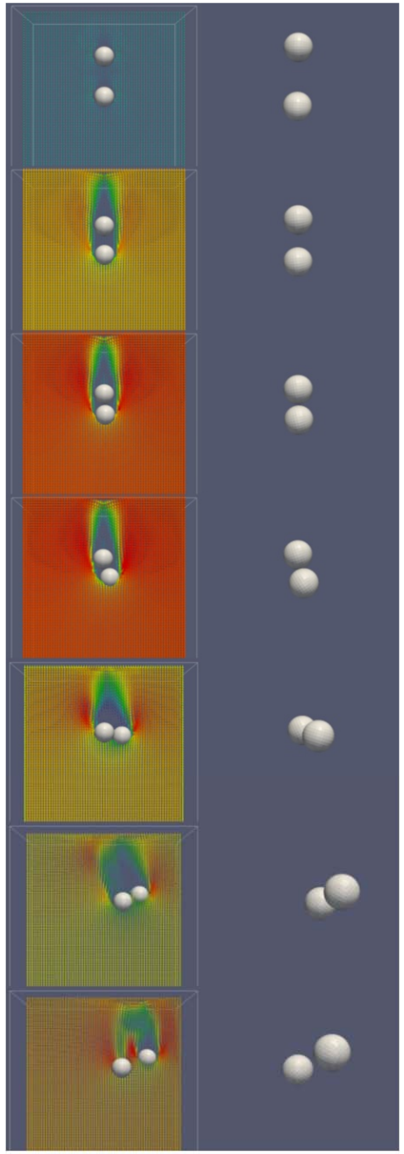
ν	$U_{featflow}$	U_{exp}	Relative error (%)
0.02	4.370	4.334	0.83
0.05	2.699	2.489	8.44
0.1	1.649	1.552	6.25
0.2	0.946	0.870	8.74

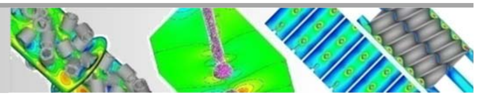
$d_s = 0.3, \quad \rho_s = 1.02$

ν	$U_{featflow}$	U_{exp}	Relative error (%)
0.01	2.167	2.107	2.84
0.02	1.495	1.436	4.11
0.05	0.809	0.749	8.01
0.1	0.402	0.404	0.44
0.2	0.218	0.216	1.02

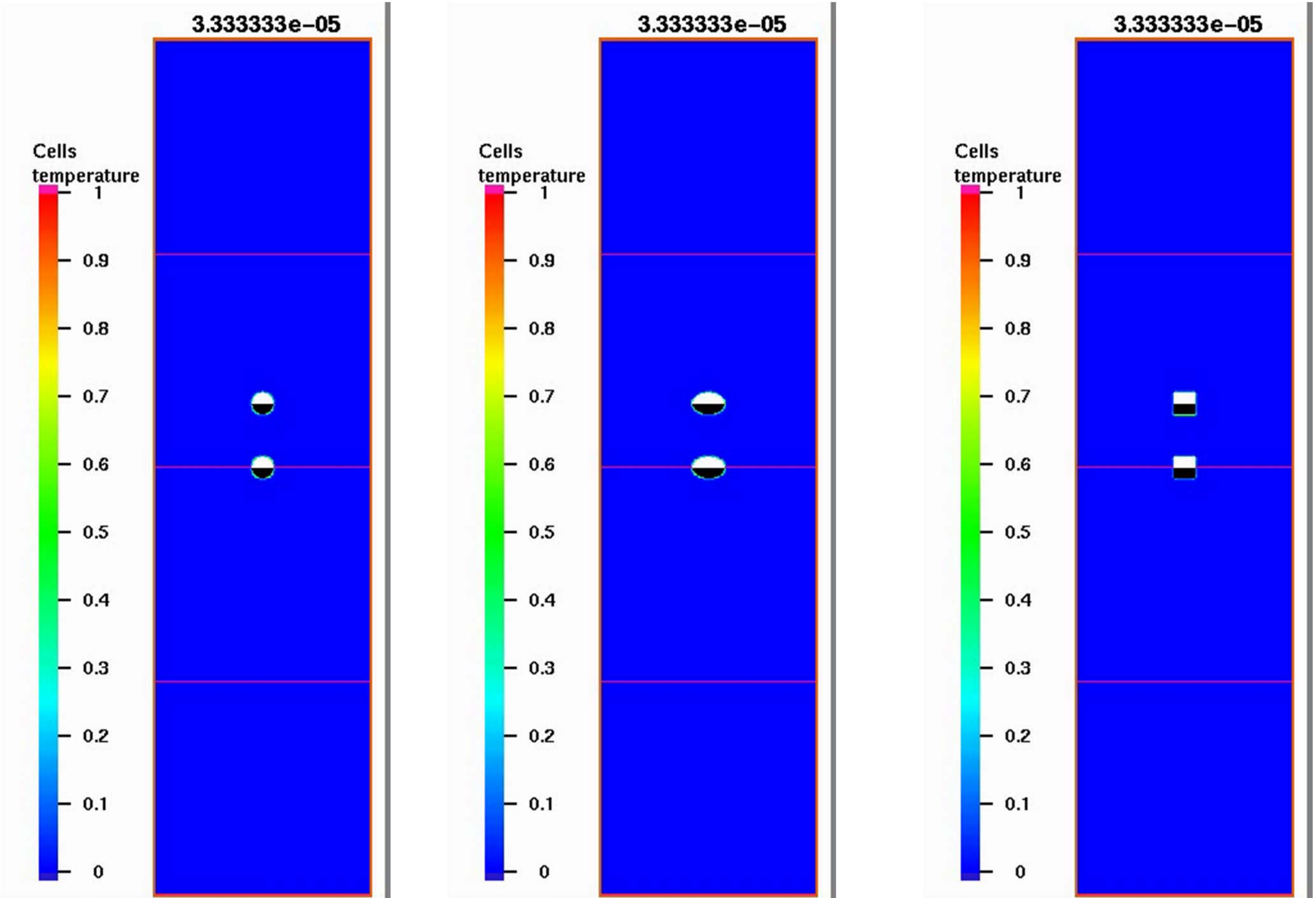
$d_s = 0.2, \quad \rho_s = 1.02$

ν	$U_{featflow}$	U_{exp}	Relative error (%)
0.01	1.4660	1.4110	3.90
0.02	0.9998	0.9129	9.52
0.05	0.4917	0.4603	6.82
0.1	0.2637	0.2571	2.57
0.2	0.1335	0.1317	1.37

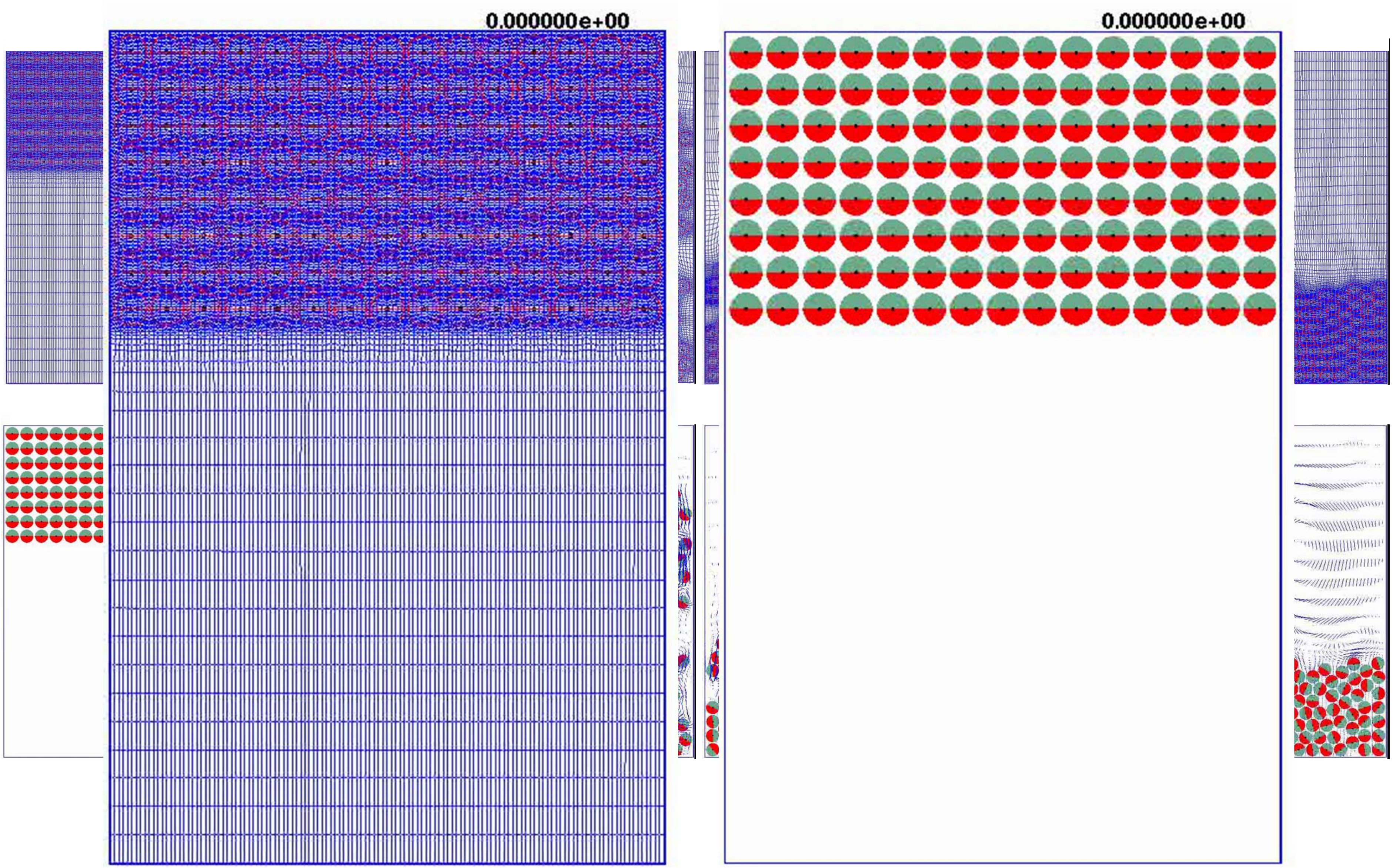




‘Kissing, Drafting, Thumbling’ of 2 Particles

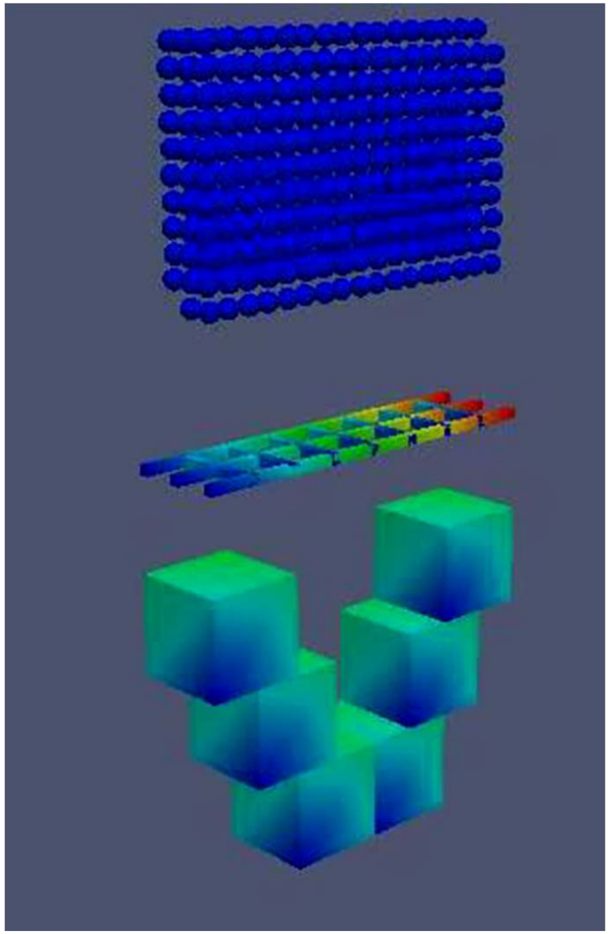


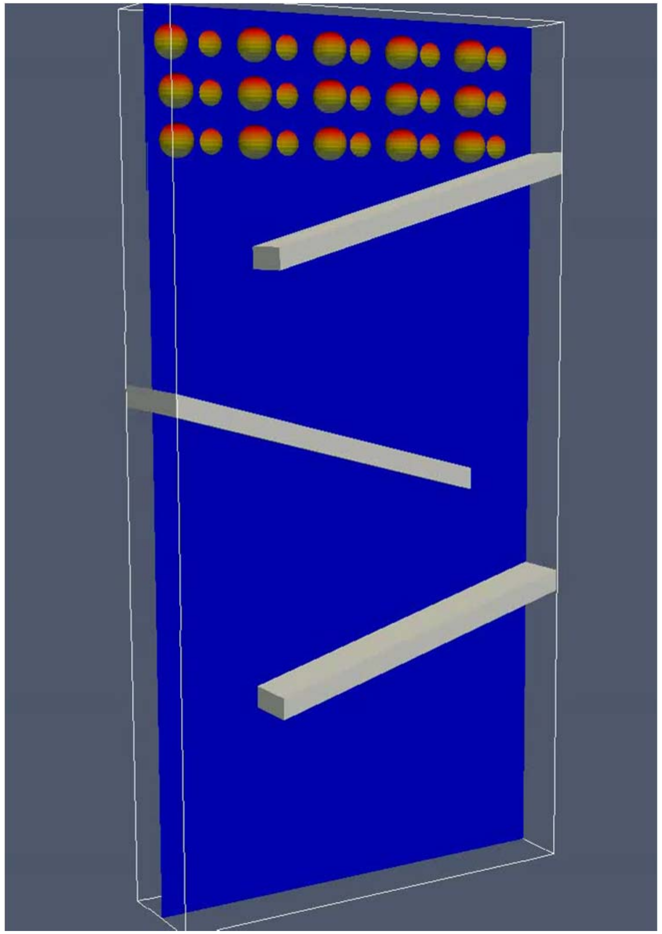
Sedimentation of many Particles



3D simulations with more complex geometries

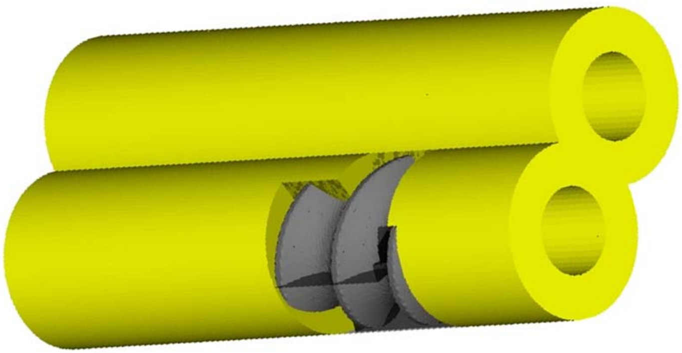
Sedimentation of particles in a complex domain

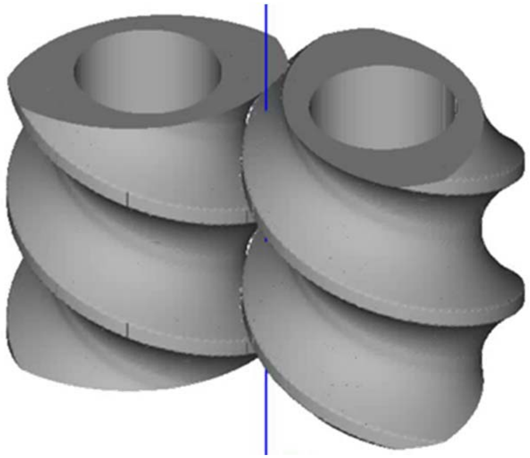




Twinscrew Flow Simulation with FBM

Geometrical representation of the twinscrews → Fictitious Boundary Method





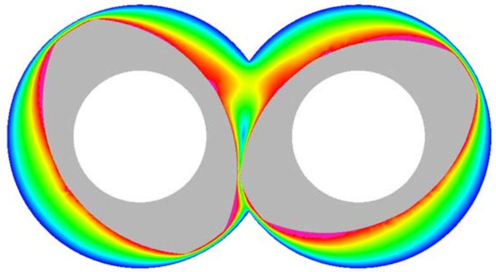
- Fast and accurate description of the rotating geometry
- Applicable for conveying and kneading elements
- Mathematical description available for
  single, double- or triplet-flighted screws
- Non-Newtonian and temperature dependent physical properties
- Heat dissipation due to high shear rates
- Viscoelastic effects and free interfaces

In cooperation with:

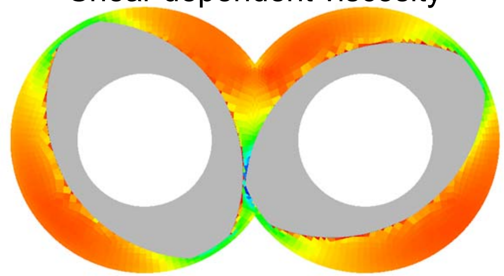
UNIVERSITÄT PADERBORN
Die Universität der Informationsgesellschaft

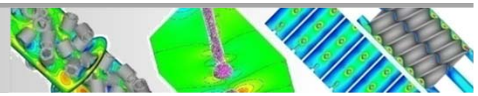
IANUS Simulation

Velocity Magnitude



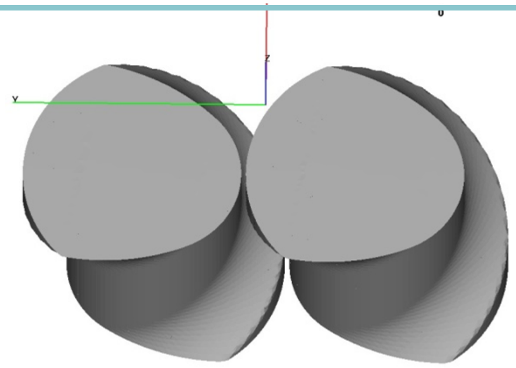
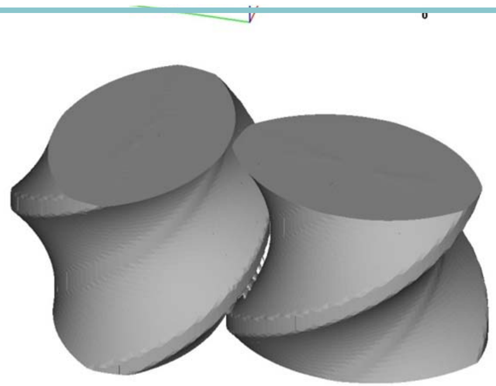
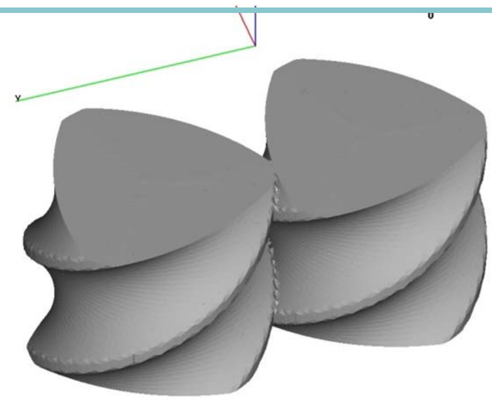
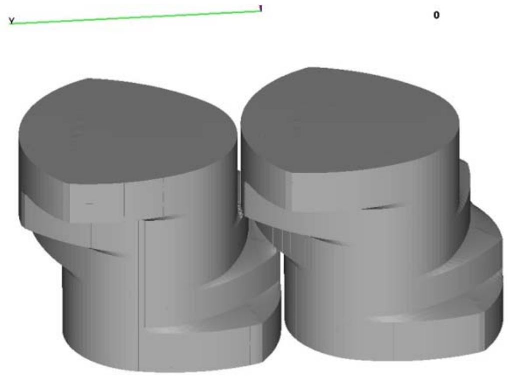
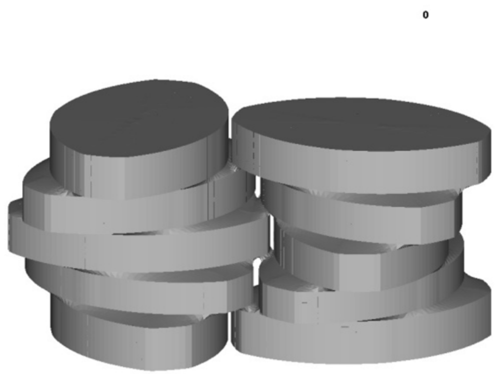
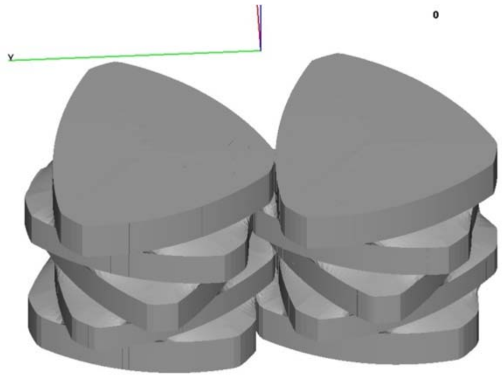
Shear dependent viscosity





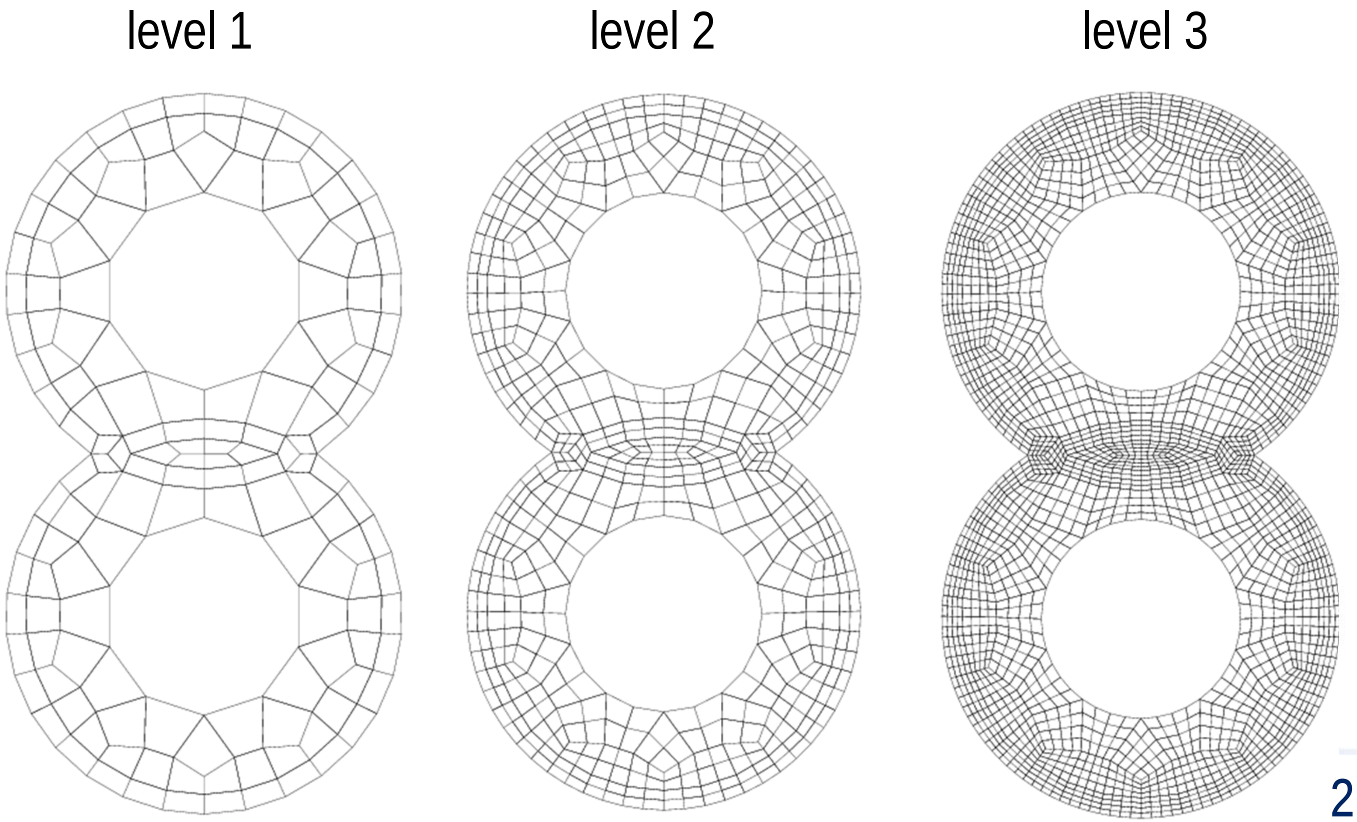
Twinscrew Flow Simulation with FBM

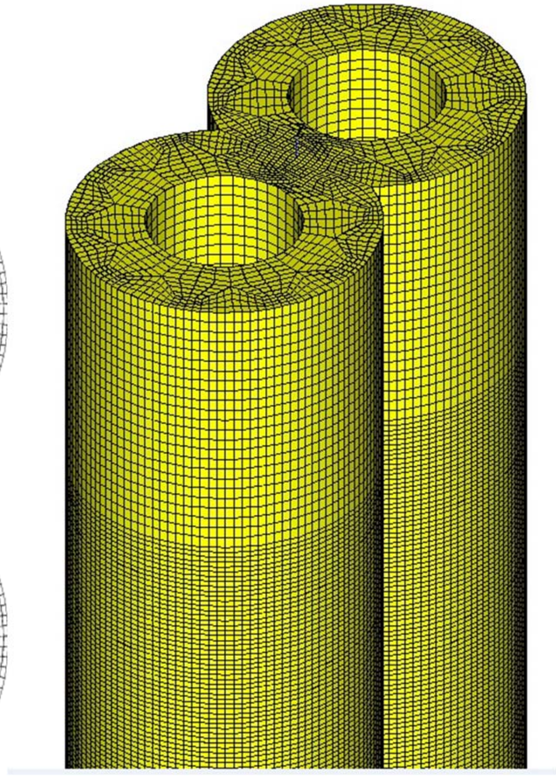
Library of conveying and mixing elements

	1 flighted	2 flighted	3 flighted
conveying			
mixing			

Twinscrew Flow Simulation with FBM

Meshing strategy – Hierarchical mesh refinement

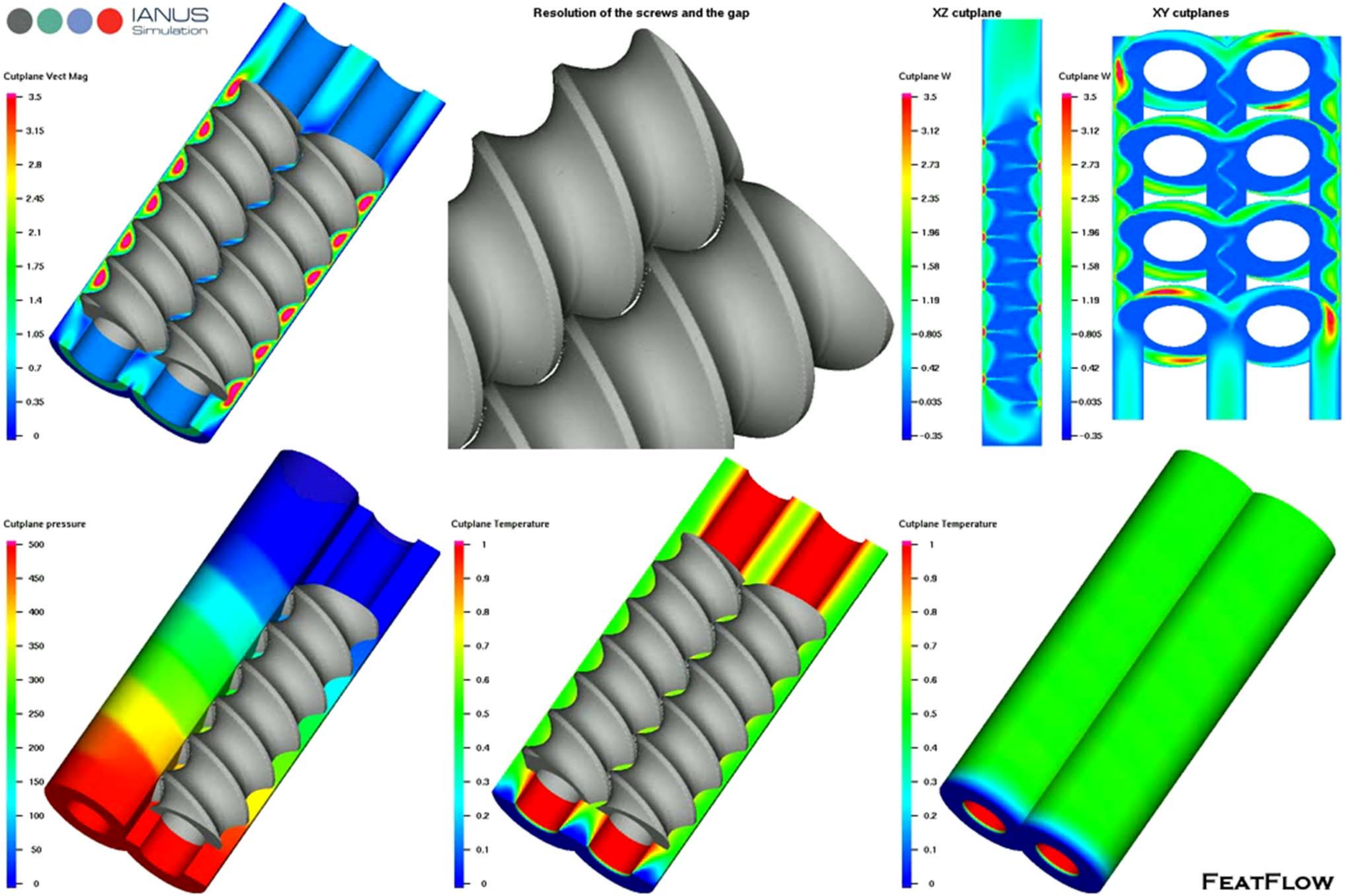




2D mesh extrusion into 3D

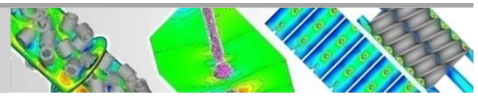
Pre-refined regions in the vicinity of gaps

Twinscrew Flow Simulation with FBM



Current Status

- **Numerical efficiency?**
  → OK
- **Parallel efficiency?**
  → OK (tested up to appr. 1000 CPUs)
  → More than 10.000 CPUs???
- **Single processor efficiency?**
  → OK (for CPU)
- **‘Peak’ efficiency?**
  → NO



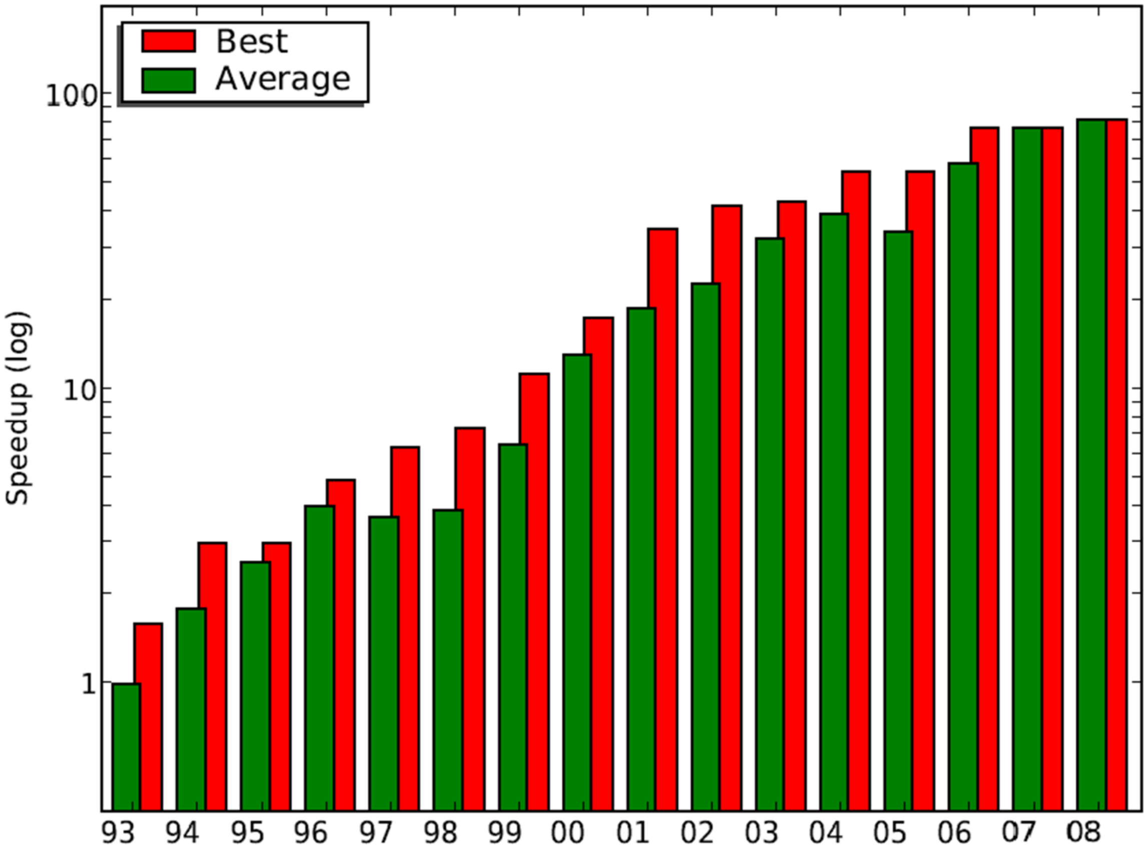
Next: Special HPC Techniques

The 'free ride' is over, paradigm shift in HPC:

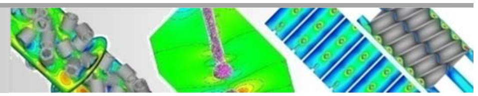
- phys ltage)
- mem oblems)
- appli er hardware

Heterogen sors

- grapl
- CELI
- HPC
- recoi



Finite Elem
solvers: mo
simulation t



Extensive Tests show…..

- It is (almost) impossible to come close to **Single Processor Peak** Performance with <u>modern</u> (= high numerical efficiency) simulation tools

- **Parallel Peak Performance** with modern Numerics even harder, already for moderate processor numbers

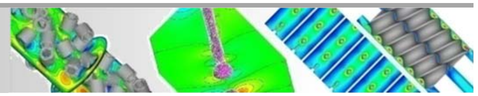
Hardware-oriented Numerics (HwoN)

+

UnConventional Hardware (UCHPC)

=

FEAST Project



Unconventional Hardware

- CELL multicore processor (PS3),
  7 synergistic processing units @ 3.2 GHz,
  Memory @ 3.2 GHz
  ≈ 218 GFLOP/s

- GPU (NVIDIA GTX 285):
  240 cores @ 1.476 GHz,
  1.242 GHz memory bus (160 GB/s)
  ≈ 1.06 TFLOP/s

UnConventional High Performance Computing (UCHPC)

Design Goals

Include GPUs into FEAST

- without
  - changes to application codes FEA(S)TFLOW
  - fundamental re-design of FEAST
  - sacrificing either functionality or accuracy

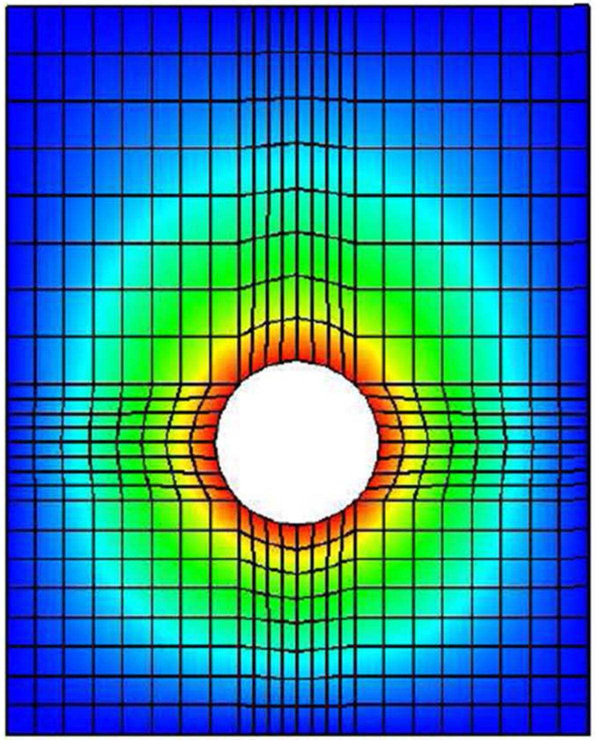
- but with
  - noteworthy speedups
  - a reasonable amount of generality w.r.t. other co-processors
  - and additional benefits in terms of space/power/etc.

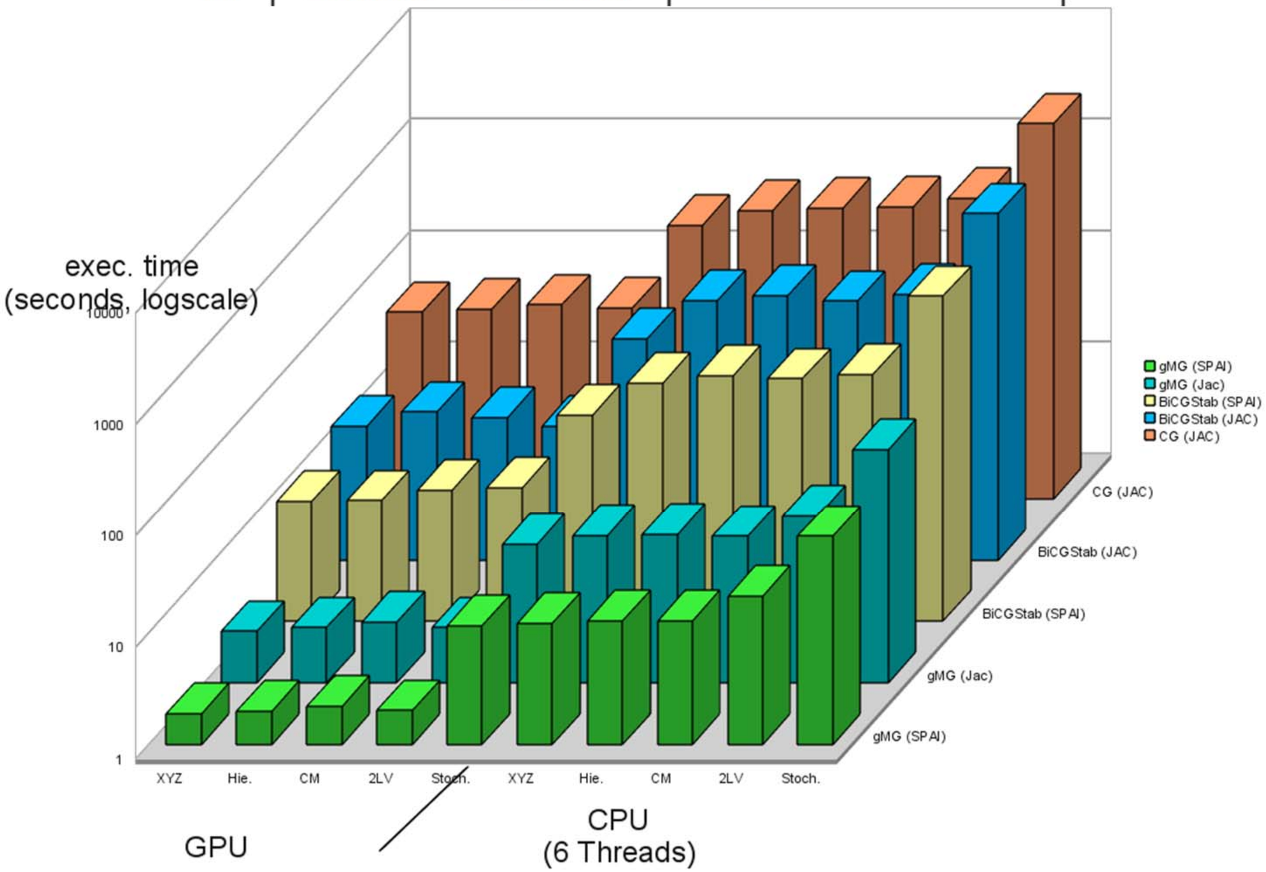
But: no --march=gpu/cell compiler switch

Solver Benchmark (unstructured mesh)

$$
\begin{aligned}
-\Delta u &= 0 && \text{in } \Omega, \\
u &= 0 && \text{on } \Gamma_1 \\
u &= 1 && \text{on } \Gamma_2
\end{aligned}
$$

L	Q_1 N	Q_1 non-zeros	Q_2 N	Q_2 non-zeros
4	576	4552	2176	32192
5	2176	18208	8448	128768
6	8448	72832	33280	515072
7	33280	291328	132096	2078720
8	132096	1172480	526336	8351744
9	526336	4704256	2101248	33480704
10	2101248	18845696	-	-

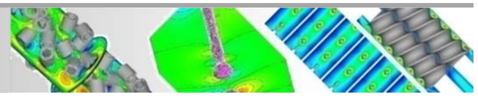




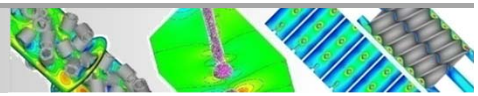
Huge Potential for the Future …

- **Numerical Simulation & High Performance Computing** have to consider recent and future **hardware trends**, particularly for heterogeneous multicore architectures and massively parallel systems!
- More research in the combination of **'Hardware-oriented Numerics'** and **'Unconventional Hardware'** is necessary!
- (Still) much more powerful CFD tools are possible if **modern Numerics meets modern Hardware**!

…or most of existing (academic/commercial) CFD software will be 'worthless' in a few years!



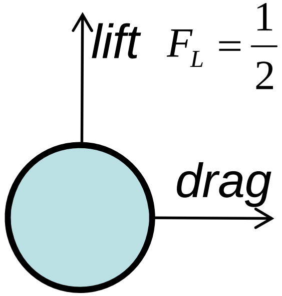
…Happy Birthday, Pekka



Benchmarking

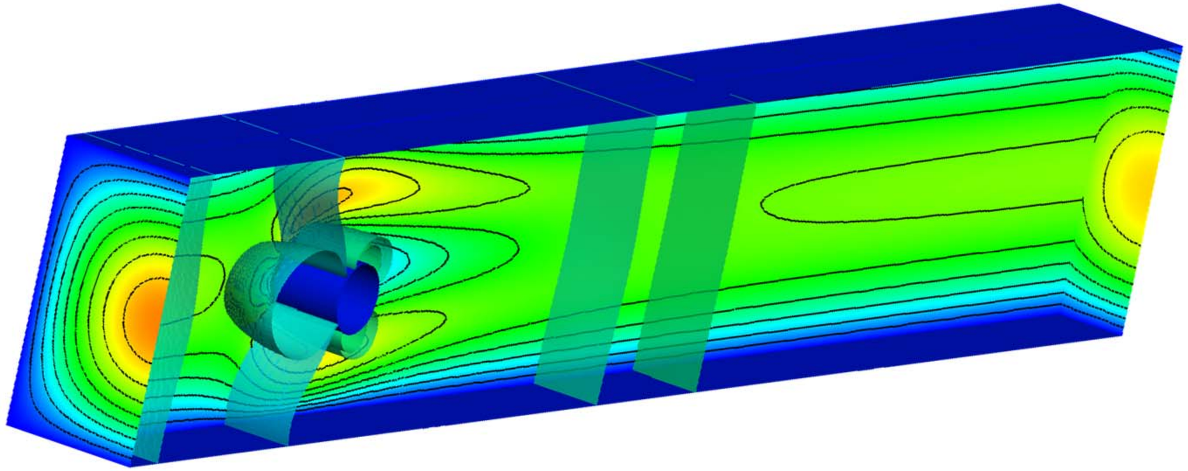
Flow Simulation with CFD software available on the market

Known benchmark problem **(DFG) in the CFD community**



$$F_L = \frac{1}{2}\rho v^2 A C_L$$

$$F_D = \frac{1}{2}\rho v^2 A C_D$$



- Comparison of CFX 12.0, **OpenFoam 1.6** and **FeatFlow**
- Drag and lift coefficients behave very sensitive to mesh resolution

→ Ideal indicator for computational accuracy

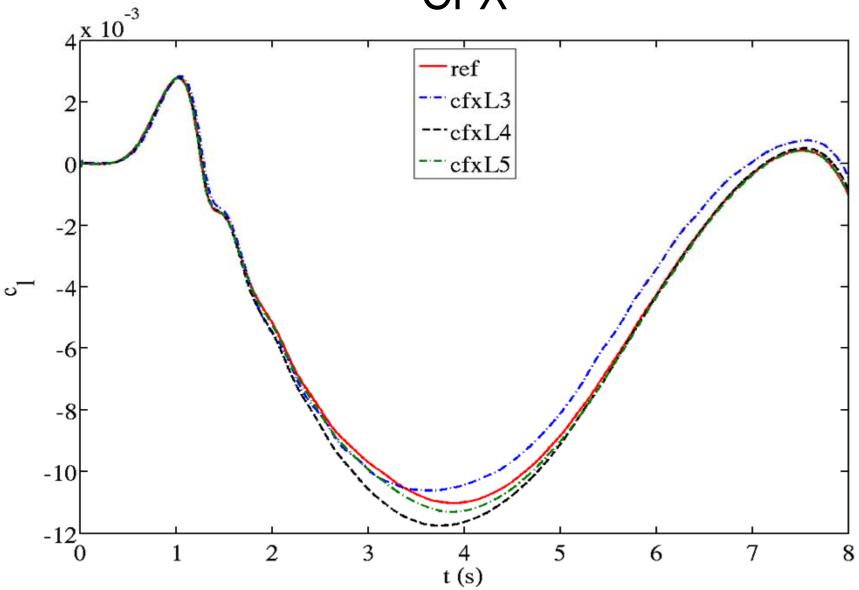
- Five consequently refined meshes L1 (coarse), …, L5 (fine)
- Same meshes and physical models used in all three codes

Mesh Level	**# Elements**
L2	6,144
L3	49,152
L4	393,216
L5	3,145,728

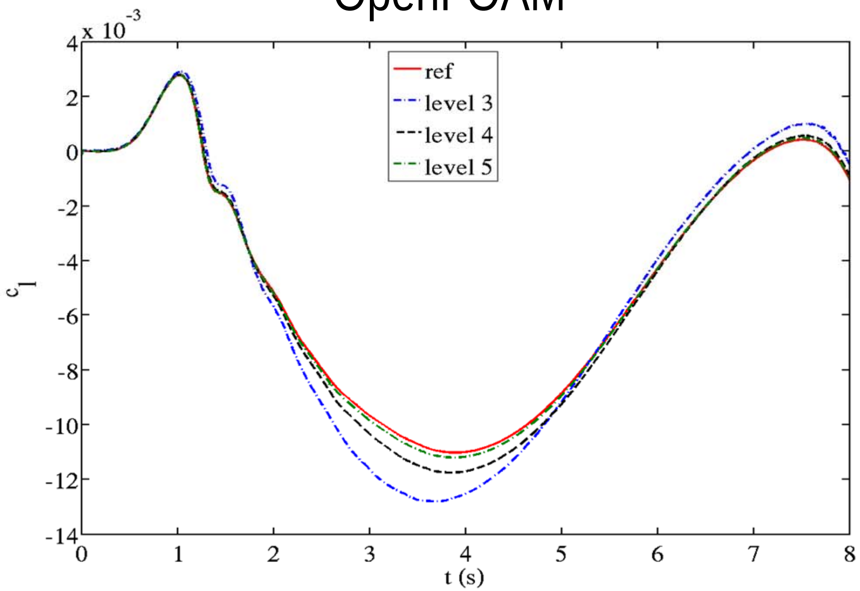
Benchmarking

Flow Simulation with CFD software available on the market

CFX

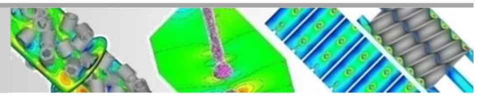


OpenFOAM



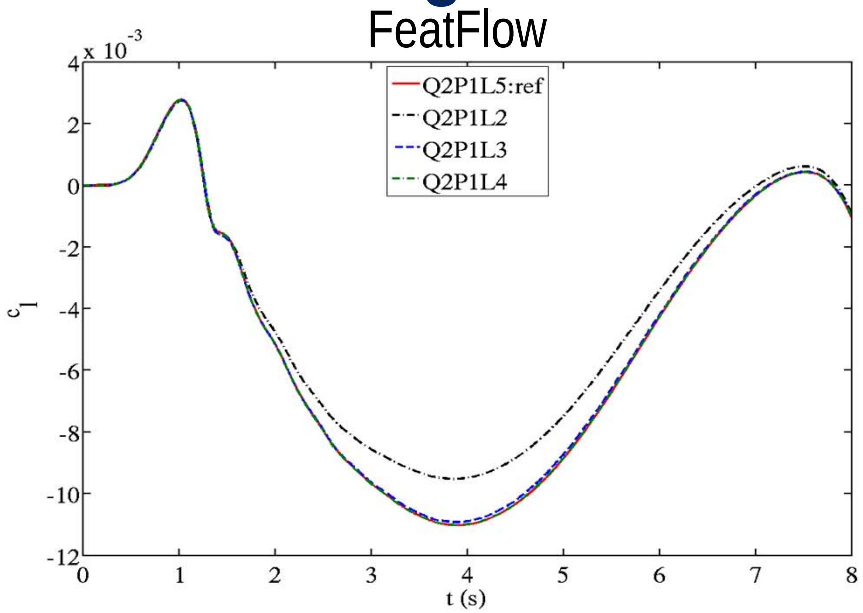
Case	L2 error		timing
	c_D	c_L	
CFX L3	0.0152	0.0781	13420
CFX L4	0.0098	0.0631	4 x 58680
CFX L5	0.0029	0.0224	8 x 205600

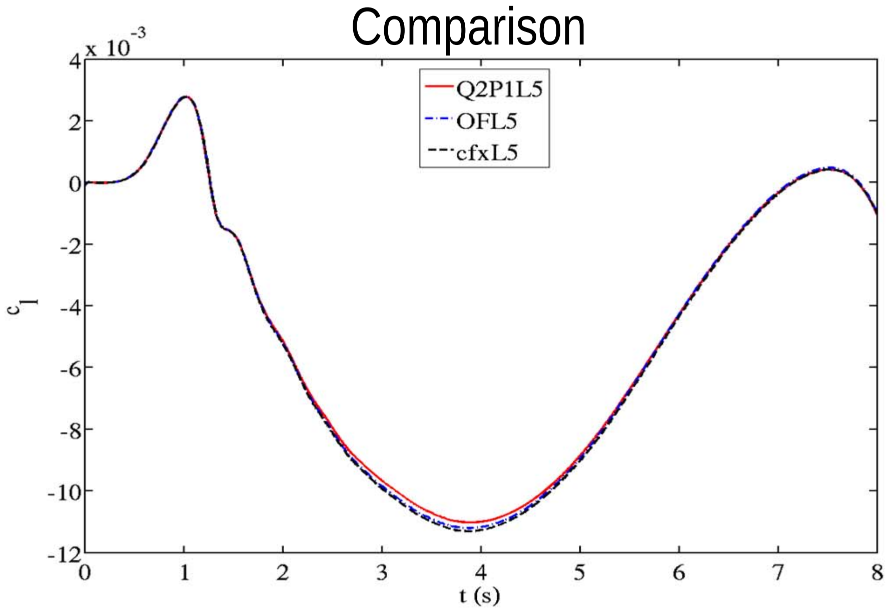
Case	L2 error		Timing
	c_D	c_L	
OF L3	0.0261	0.1449	5180
OF L4	0.0067	0.0591	4 x 19500
OF L5	0.0016	0.0147	8 x 595200



Benchmarking

Flow Simulation with **FEATFLOW**





Case	L2 error		Timing
	c_D	c_L	
FF L2	0.0209	0.1378	2 x 5000
FF L3	0.0029	0.0109	3 x 25000
FF L4	0.0005	0.0015	20 x 32000
FF L5	(ref)	(ref)	23 x 242000

Case	L2 error		timing
	c_D	c_L	
FF L3	0.0029	0.0109	3 x 25000
OF L5	0.0016	0.0147	8 x 595200
CFX L5	0.0029	0.0224	8 x 205600

Less than 2 hours sim. time with adaptive time stepping on 3+1 processors

→ Same order of accuracy with **FEATFLOW** on L3 as L5 with CFX and OpenFOAM on L5!

→ High order Q2/P1 FEM + (parallel) Multigrid Solver

Validation based on experimental results

Jetting mode

Experimental setup/results by **AG Walzel** **(BCI/Dortmund)**

Continuous phase:

Glucose-Water mixture

$\mu_D = 500\, mPa\, s$

$\rho_D = 972\, kg\, m^{-3}$

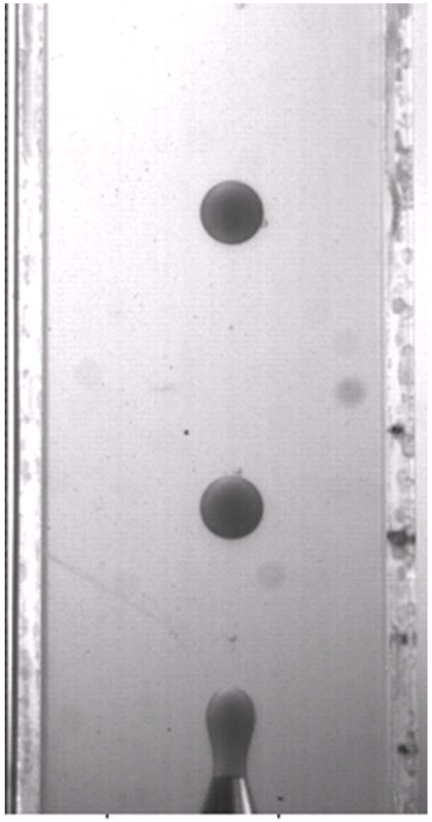
$\sigma_{CD} = 0{,}034\, N\, m^{-1}$

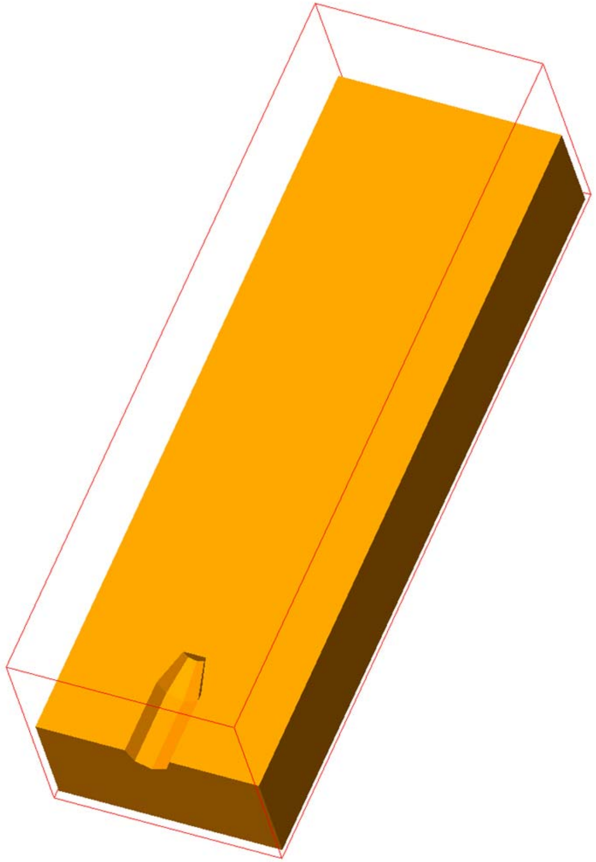
Silicon oil

$\mu_C = 500\, mPa\, s$

$\rho_C = 1340\, kg\, m^{-3}$

Dispersed phase:



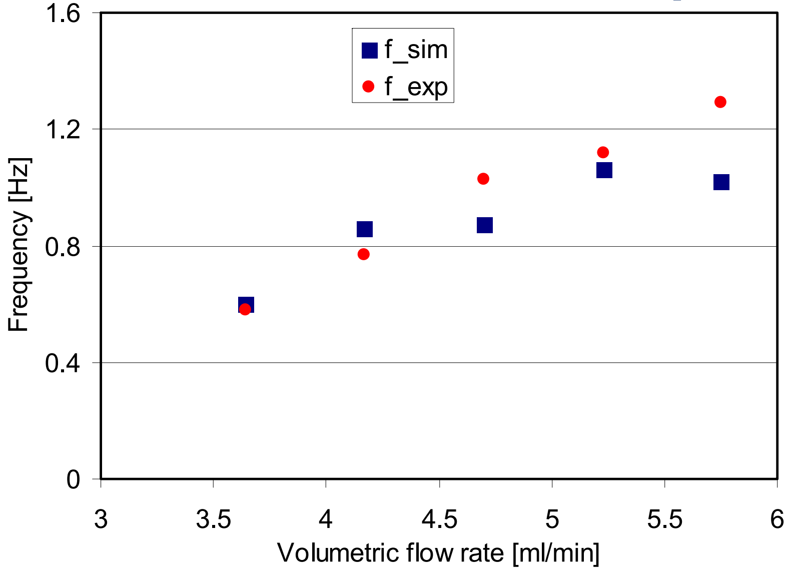


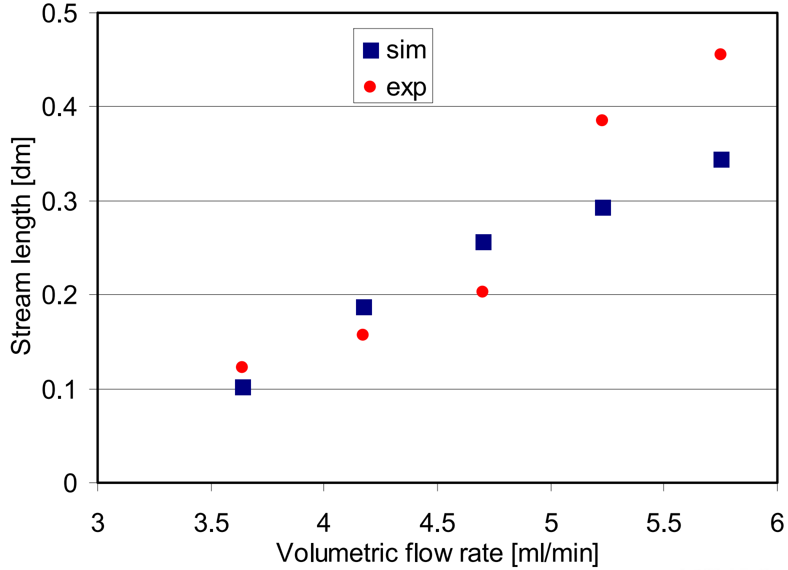
Operating conditions

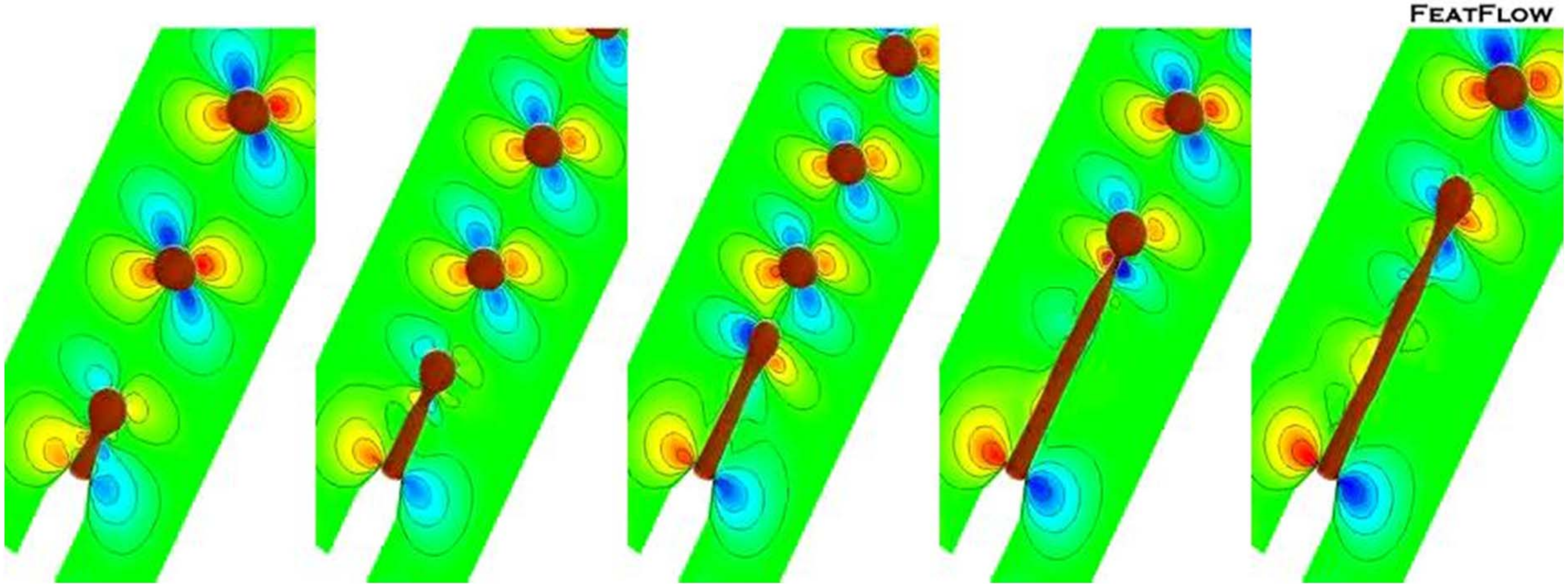
V_D [ml/min]	3,64	4,17	4,70	5,23	5,75
V_C [ml/min]	99,04	113,34	128,34	143,34	156,95

Validation parameters:
- frequency of droplet generation
- droplet size
- stream length

Validation based on experimental results







3,64 ml/min   4,17 ml/min   4,70 ml/min   5,23 ml/min   5,75 ml/min