Thin Film Flow Simulation on a Rotating Disc

Strömungsforschung GmbH

ICE

Innovative Computational Engineering

P. Vita, B. Gschaider, D. Prieling, H. Steiner

Overview

- **Introduction**
  - Problem description
  - OpenFOAM
  - Finite Area Method
- **Model development**
  - Thin Film Model
  - Impinging Jet
  - Polydual Mesh
- **Results**
  - Comparison with 3D Solution
- **Conclusion/Discussion**

Motivation

- **Our industry partner, LAM Research AG, initiated a project to be able to optimize they product, a spin processor**
  - One-sided single wafer wet processing
  - Patented wafer chuck with floating wafer (N_2 cushion)
  - Vertically arranged process levels
  - Clearly separated chemical lines

Motivation - State of the Art in Models

- **2D Simulation (Axial-Symmetric)**
  - Advantages
    - Reasonably small meshes
    - Short computation times in order of hours
    - No additional model assumptions
  - Disadvantages
    - Allows only central impingement
    - Resolve waves only in radial direction

- **3D Simulation**
  - Advantages
    - Fine resolution only where required (with adaptive mesh refinement)
    - No additional model assumptions
  - Disadvantages
    - Huge meshes
      - Still cannot fully resolve all physical aspects
    - Long computation times in order of weeks/months

OpenFOAM

- **OpenFOAM is a free, open source CFD software package**
  - C++ toolbox for development of custom numerical solvers, pre- and post-processing utilities
  - contains a many CFD solvers
    - compressible, incompressible, RANS, LES, multi-phase flows, particle tracking, combustion, conjugate heat transfer etc.
  - Finite Volume Method
  - arbitrary polyhedral meshes
  - support for parallel processing
  - “official” branch by H. Weller, ~~SGI Corp~~ ESI
  - “extended” branch by H. Jasak, University of Zagreb

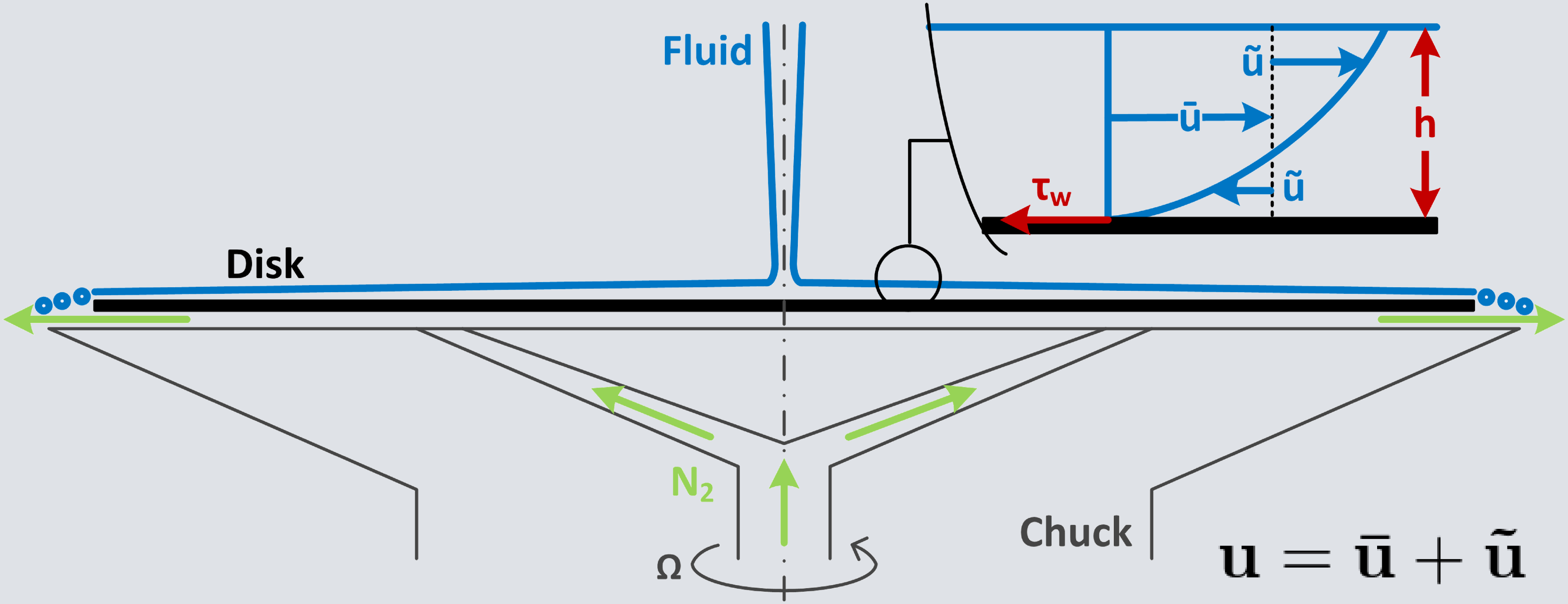
Finite Area Method

- **Specialization of FVM to flows on surfaces-films**
  - Takes surface curvature into account
- **Implementation by H. Jasak and Z. Tukovic in OpenFOAM-ext project**
  - Only present in 1.5-dev and 1.6-ext version
- **Demonstration solver models the transport equation on a prescribed velocity field**
  - `surfactantFoam` solver
- **Equations are solved on a boundary patch of the volume mesh**
  - FV-solution can be used as a source term

Thin Film Model - Assumptions

- Normal velocity component is negligible compared to tangential one
- Pressure gradient is constant across the film thickness
- Laminar flow
- Air/liquid shear stress interactions at the film surface are neglected
- Parabolic velocity profile assumed across the film thickness
- Gravity acts against the disk normal direction

Thin Film Model - Rotating Disk Scheme



$$\mathbf{u} = \bar{\mathbf{u}} + \tilde{\mathbf{u}}$$

$$\bar{\mathbf{u}} = \frac{1}{h} \int_h \mathbf{u}\, dz$$

- **Dependent variables**
  - Film thickness *h*
  - Mean velocity $\bar{\mathbf{u}}$

Thin Film Model - Conservation Eqs.

- **Continuity Equation**

$$\frac{\partial h}{\partial t} + \nabla\bullet\left(h\overline{\mathbf{u}}\right) = S_m$$

- **Momentum Equation**

$$\frac{\partial}{\partial t}\left(h\overline{\mathbf{u}}\right) + \nabla\bullet\left(h\overline{\mathbf{u}\mathbf{u}} + \mathbf{C}\right)$$

$$= -\frac{1}{\rho}h\nabla\left(\rho|\mathbf{g}|h + \sigma\nabla\bullet\nabla h\right) - \frac{1}{\rho}\tau_{\mathrm{disk}} + \mathbf{S}_m$$

Thin Film Model - Velocity Profile

- **In order to describe the shear stress at the disk and the differential advection, we introduce a polynomial velocity profile function**

$$\mathbf{u}(x,y,z) = u(x,y,\xi) + \varepsilon_u$$

$$u(x,y,\xi) = \mathbf{a}_0 + \mathbf{a}_1\xi + \mathbf{a}_2\xi^2 + \mathbf{a}_3\xi^3$$

$$\xi \in \langle 0,1 \rangle, z = h\xi$$

- **where ε_u represents the modelling error and ξ is a normalised vertical coordinate**

Thin Film Model - Velocity Profile Boundary Conds.

- **and fulfils the following boundary conditions**

$$\int_0^1 u(\xi)\, d\xi = \bar{\mathbf{u}}$$

$$u(\xi)|_{\xi=0} = \mathbf{u}_{\text{disk}}$$

$$\left.\frac{\partial u(\xi)}{\partial \xi}\right|_{\xi=1} = 0$$

$$\left.\frac{\partial^2 u(\xi)}{\partial \xi^2}\right|_{\xi=0} = 0$$

Thin Film Model - Diff. Advection and Shear Stress

Strömungsforschung GmbH

Innovative Computational Engineering

- The boundary conditions lead to the following differential advection solution

$$\mathbf{C} = \int_h \tilde{\mathbf{u}}\tilde{\mathbf{u}}\, dz = \left[\frac{213}{875} h \left(\overline{\mathbf{u}} - \mathbf{u}_{\text{disk}}\right)\left(\overline{\mathbf{u}} - \mathbf{u}_{\text{disk}}\right)\right]$$

- and the shear stress at the disk

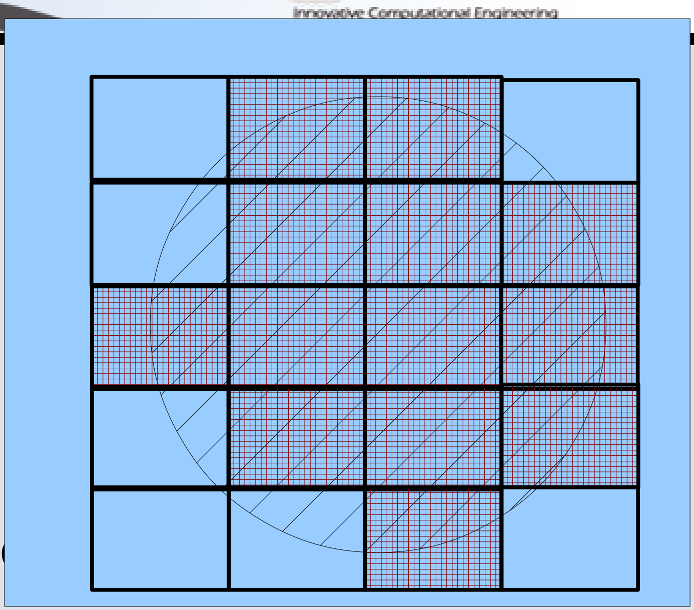
$$\tau_{\text{disk}} = \mu \left.\frac{\partial \mathbf{u}}{\partial z}\right|_{z=0} = \frac{\mu}{h}\frac{12}{5}\left(\overline{\mathbf{u}} - \mathbf{u}_{\text{disk}}\right)$$

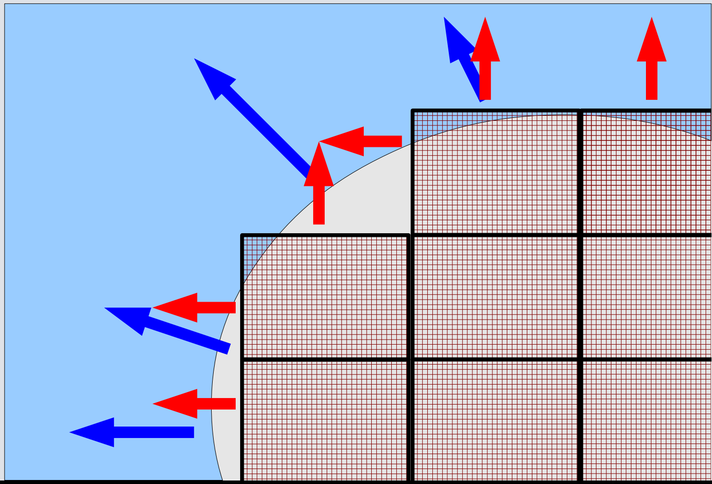
Impinging Jet

- **Impingement area is generally not know**
  - – Impinging jet is moving over the disk
- **Thin film model is not valid in the impingement area and its surrounding**
  - – Solution in the impingement area is known from FVM
  - – Impingement area is “weakly” influenced from “outside”
- **Possible impingement implementations**
  - – Remeshing
    - Impingement area is represented by a circular boundary condition which moves and the mesh is adapted
  - – Fixation of solution in faces
    - Impingement faces are selected and solution is prescribed

Impinging Jet

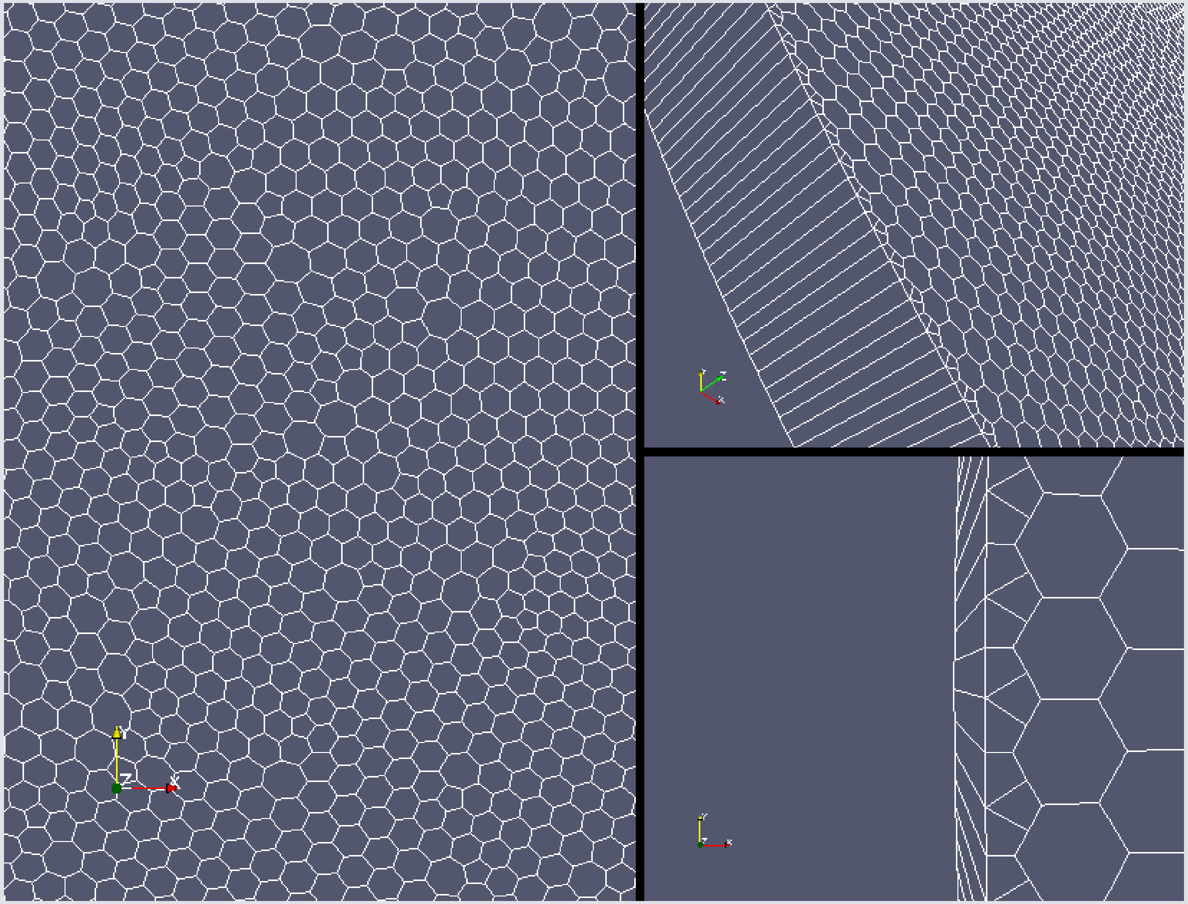
- **Fixation of solution in the faces has significant advantages over remeshing, however it has its own problems**
  - "Crown Cap" effect
    - Faces in the impingement area are n
      resolving exact circle
    - Face boundaries are not aligned with
      circle
  - Total mass-flow correction
  - Inlet velocity profiles
    - Velocities varies along the jet edge





Polydual Mesh

- **Solution is very mesh sensitive**
  - Mesh neutral to flow is needed to avoid artefacts
    - “flow arms”
    - “rose petals”
  - Polyhedral mesh shown the best results
    - **`polyDualMesh`** utility used to convert a tetrahedral mesh into the polyhedral one

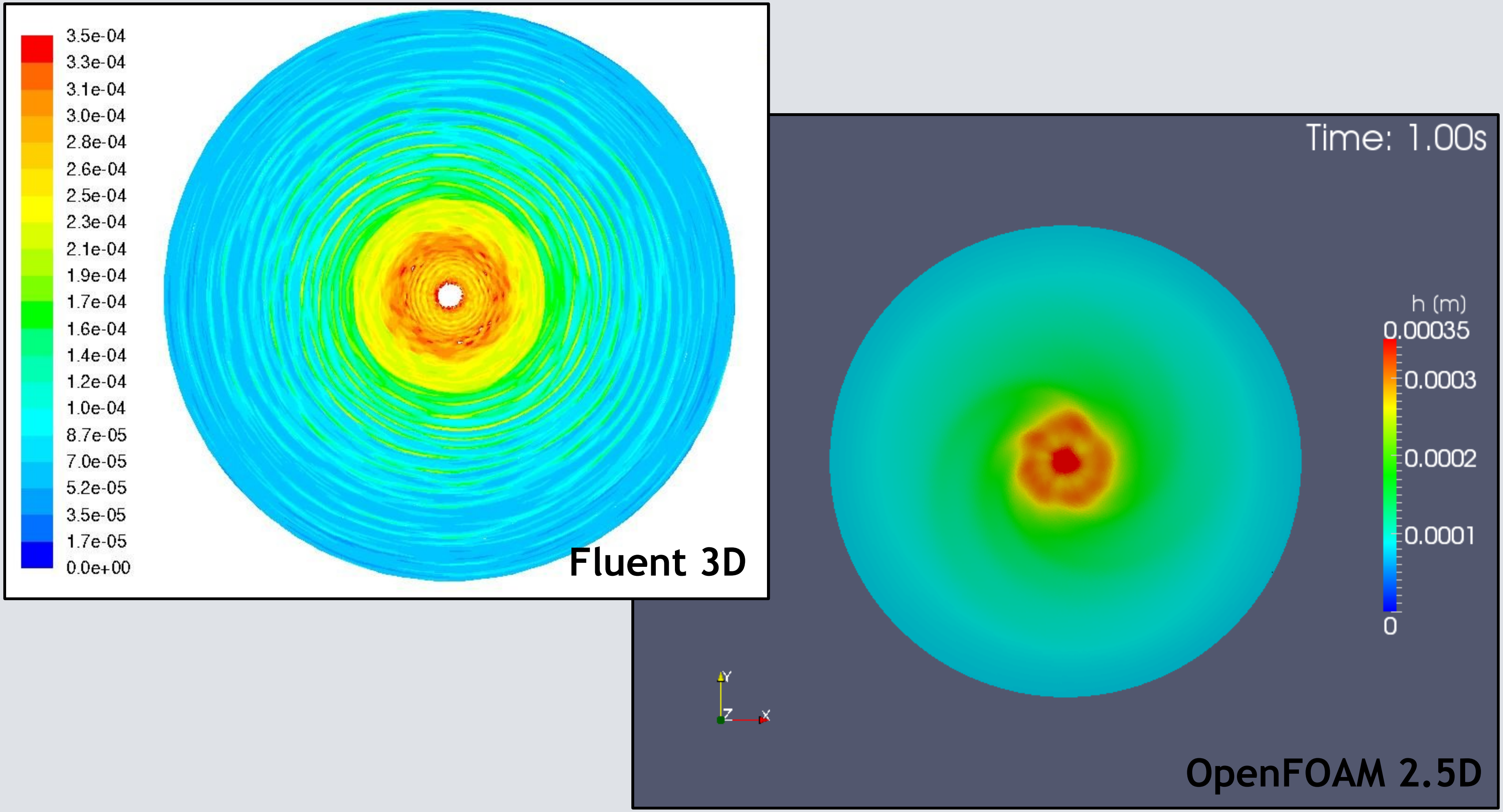


Comparison with 3D Solution

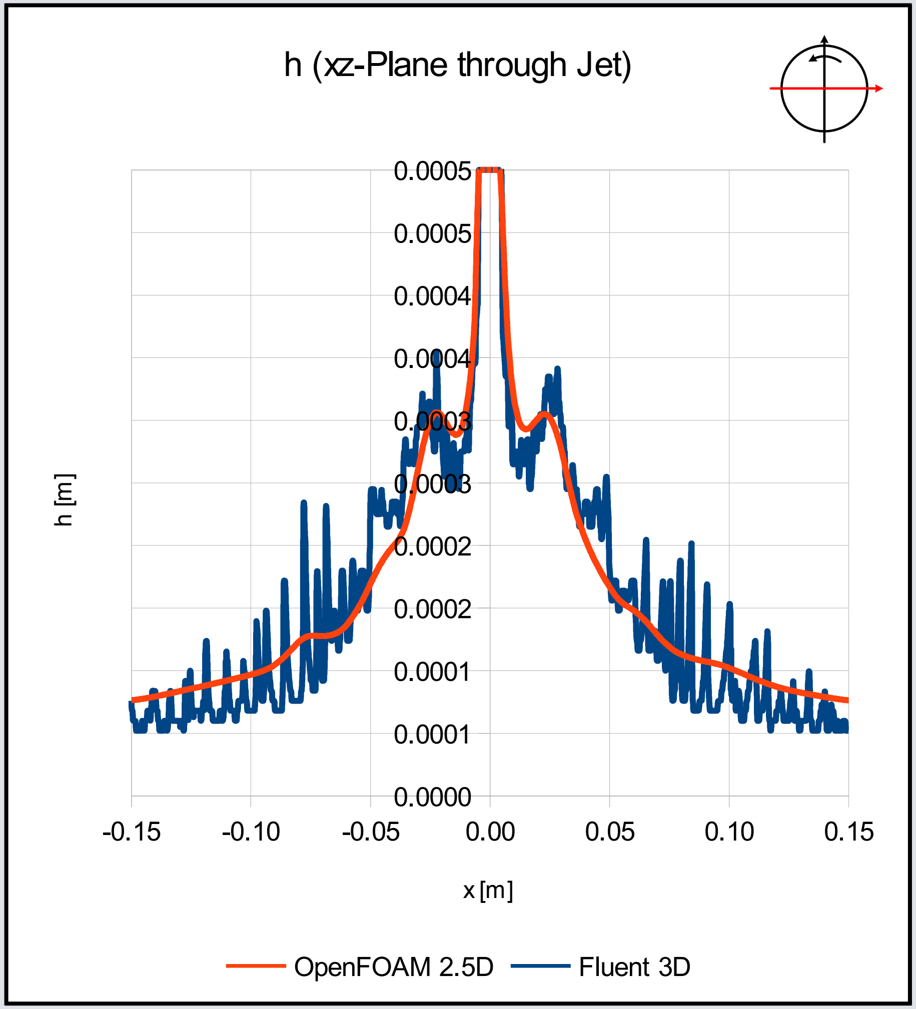
- **3D solution**
  - Fluent software
  - 5M cells, 4 CPU cores used
  - **1s of process ~ 30days**
- **2.5D solution**
  - OpenFOAM software
  - 36.8k polydual mesh, single CPU core used
  - **1s of process ~ 2hours**

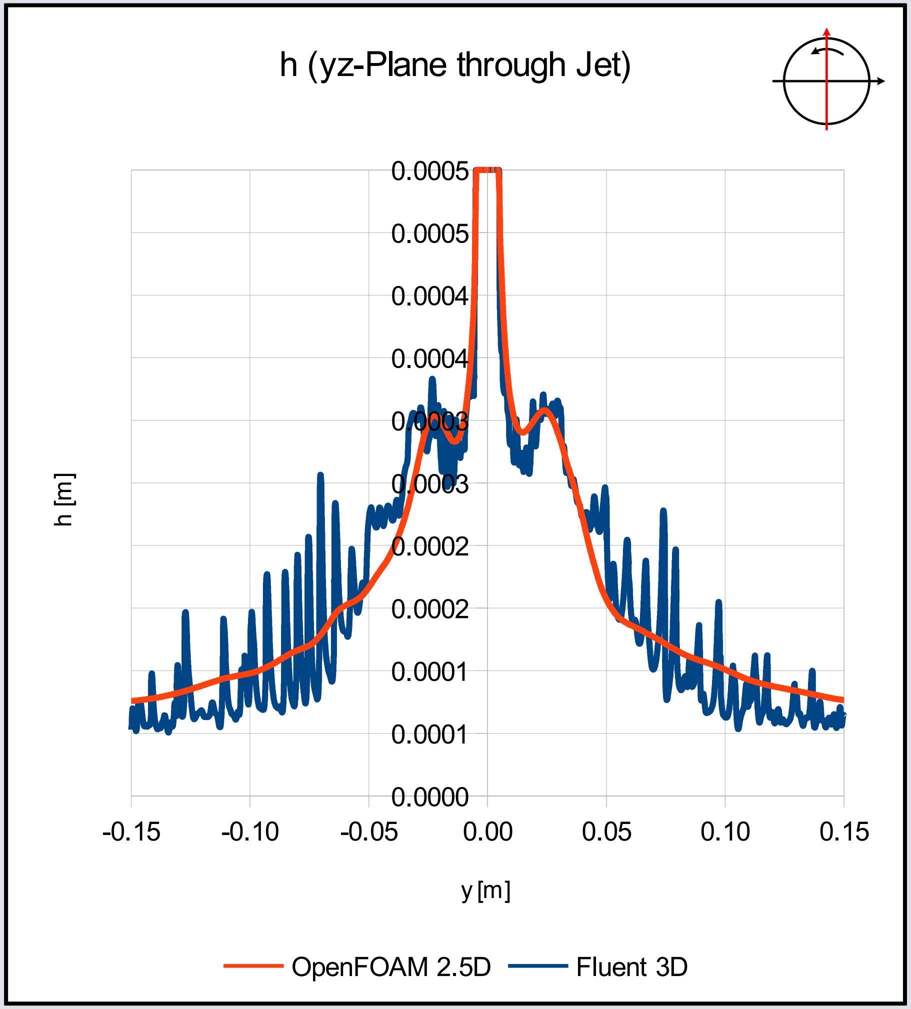
- **Cases**
  - $\Omega = 500$rpm, $Q = 1.5$l, Spinetch-D ($\nu = 2.87\times10^{-6}$)
  - Impingement area
    - Reference Case (central impingement)
    - Case 1a (ex-centric case, $\Delta r = 30$mm)
  - **No moving inlet due to 3D solution limitation**

Reference Case: 500rpm, 1.5lpm, Spinetch-D

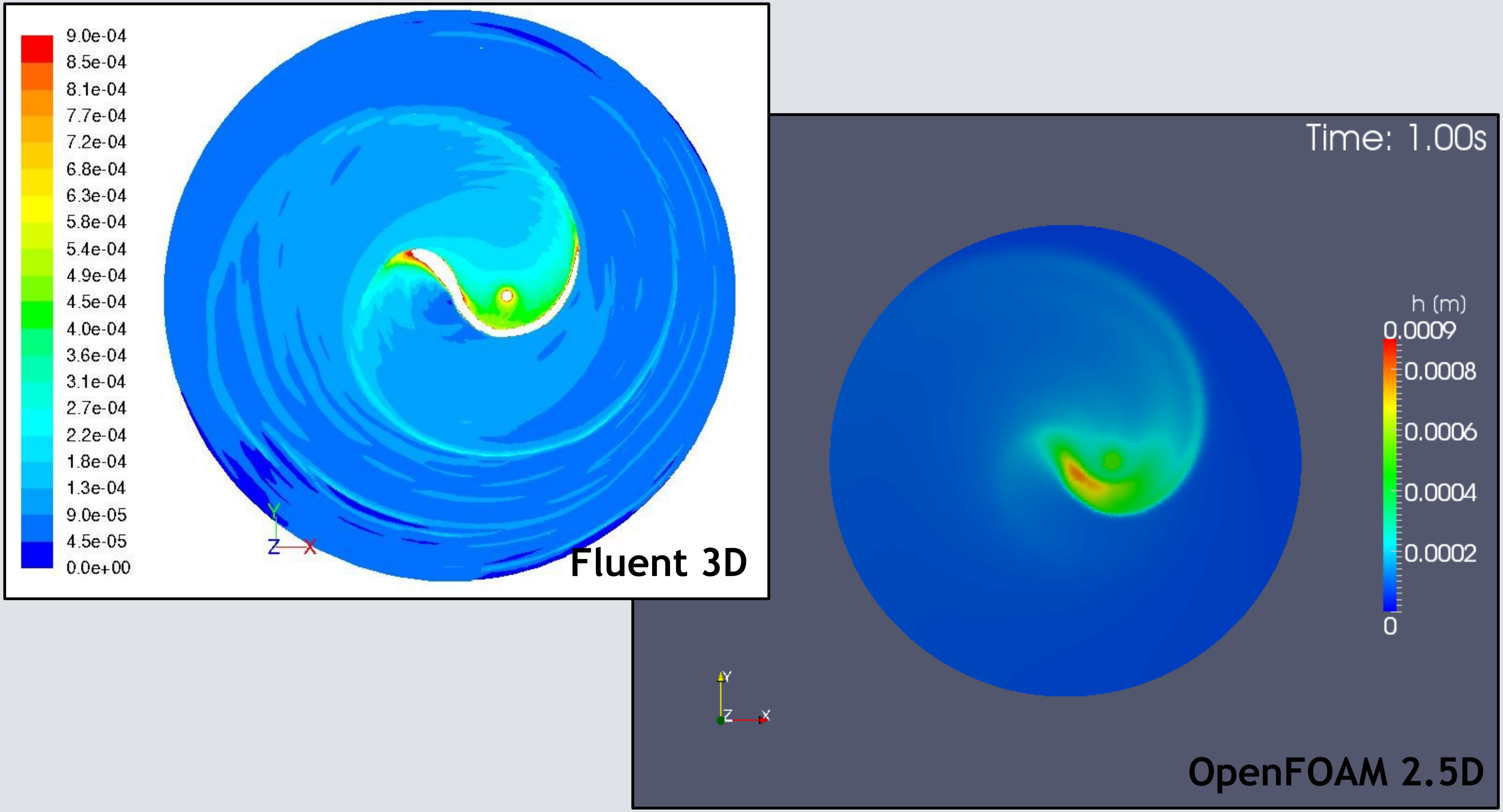


Reference Case: 500rpm, 1.5lpm, Spinetch-D

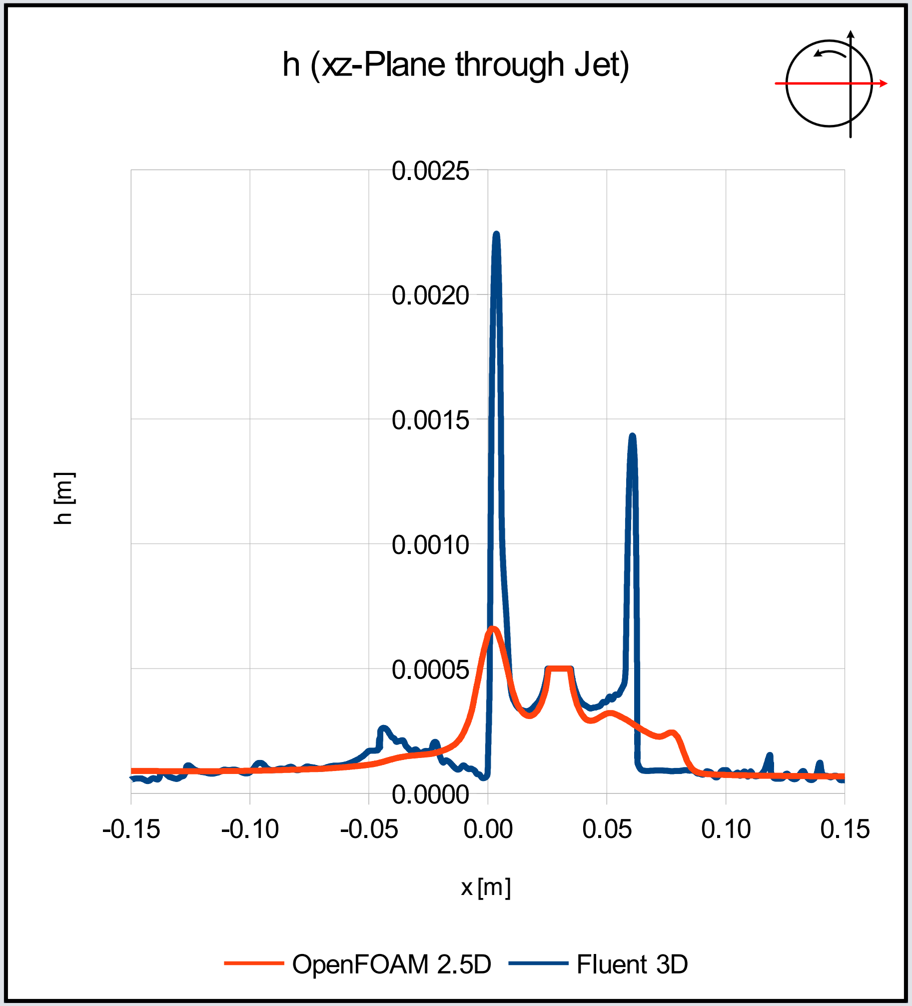


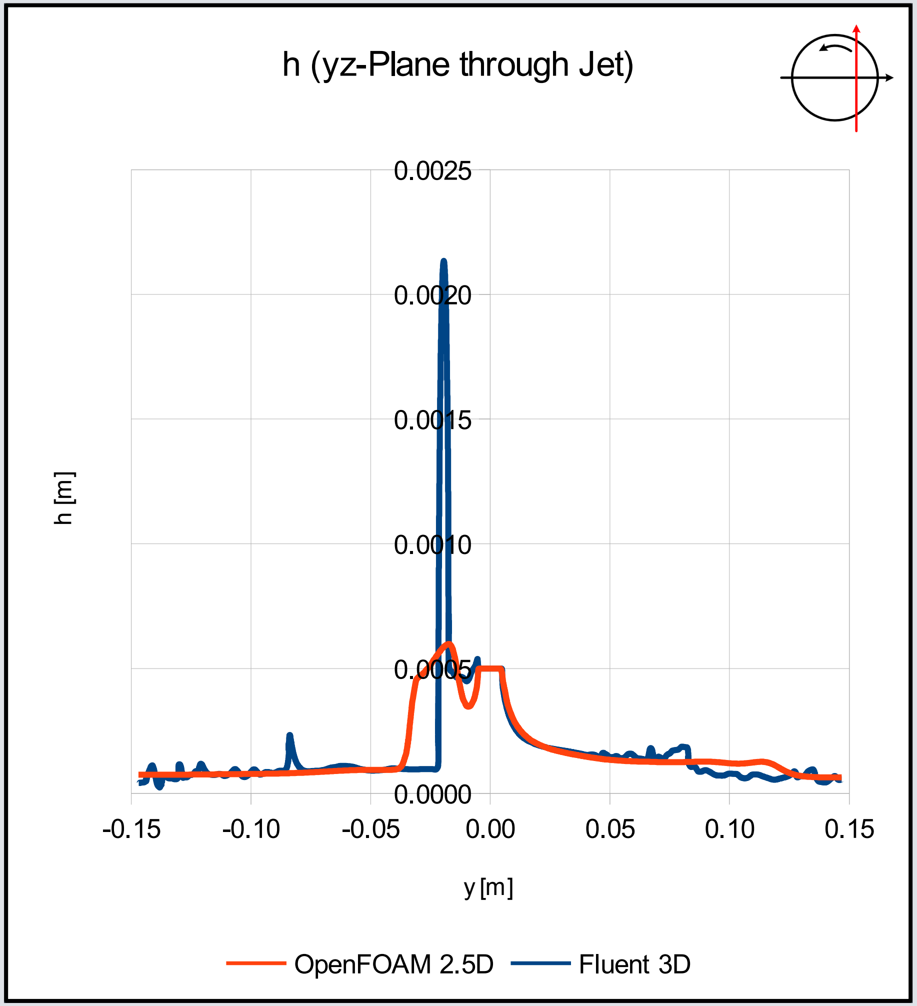


Case 1a: 500rpm, 1.5lpm, Δr=30mm, Spinetch-D

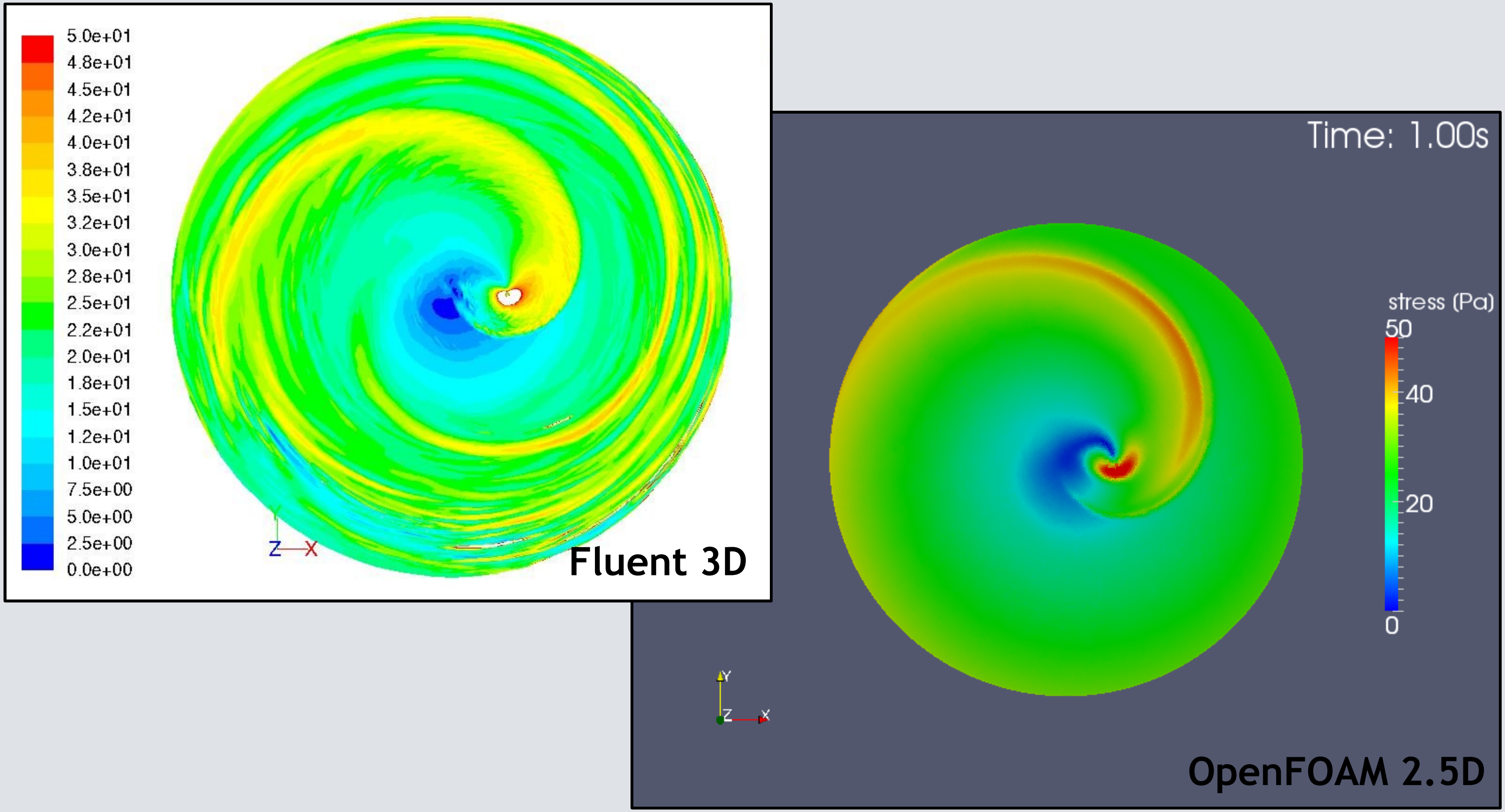


Case 1a: 500rpm, 1.5lpm, Δr=30mm, Spinetch-D

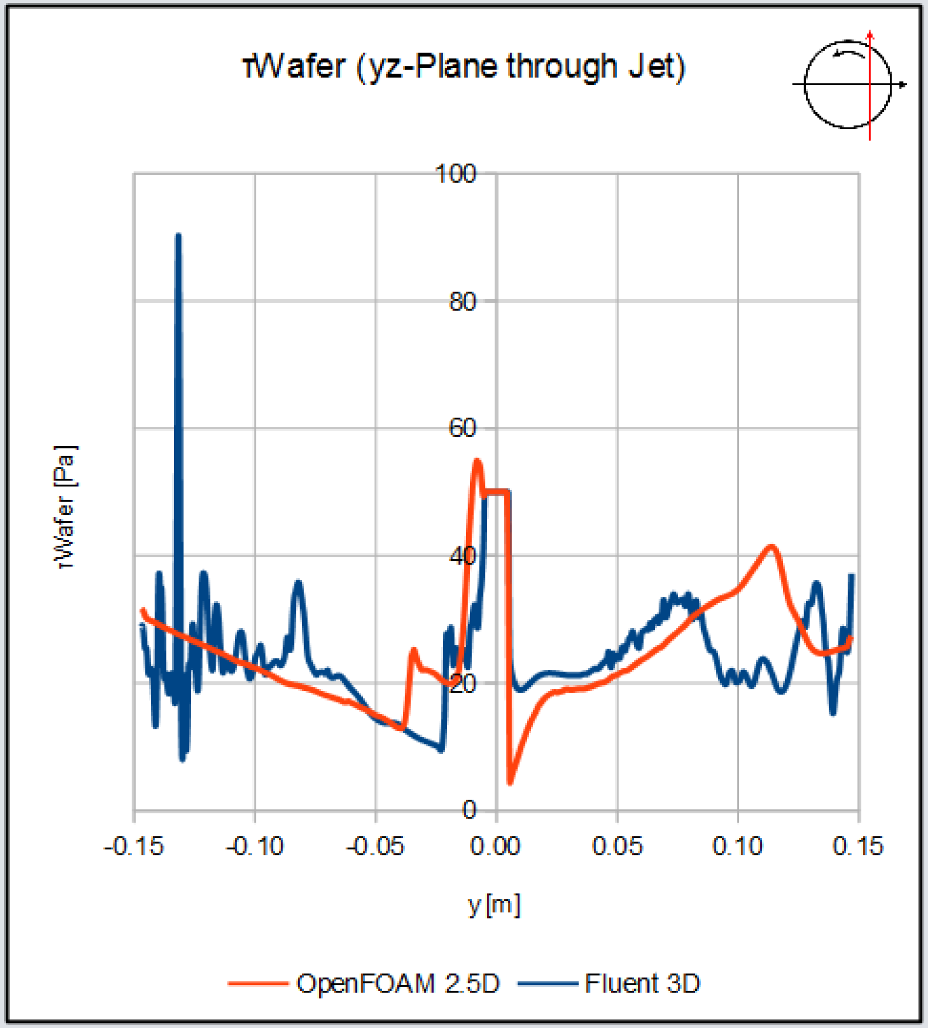


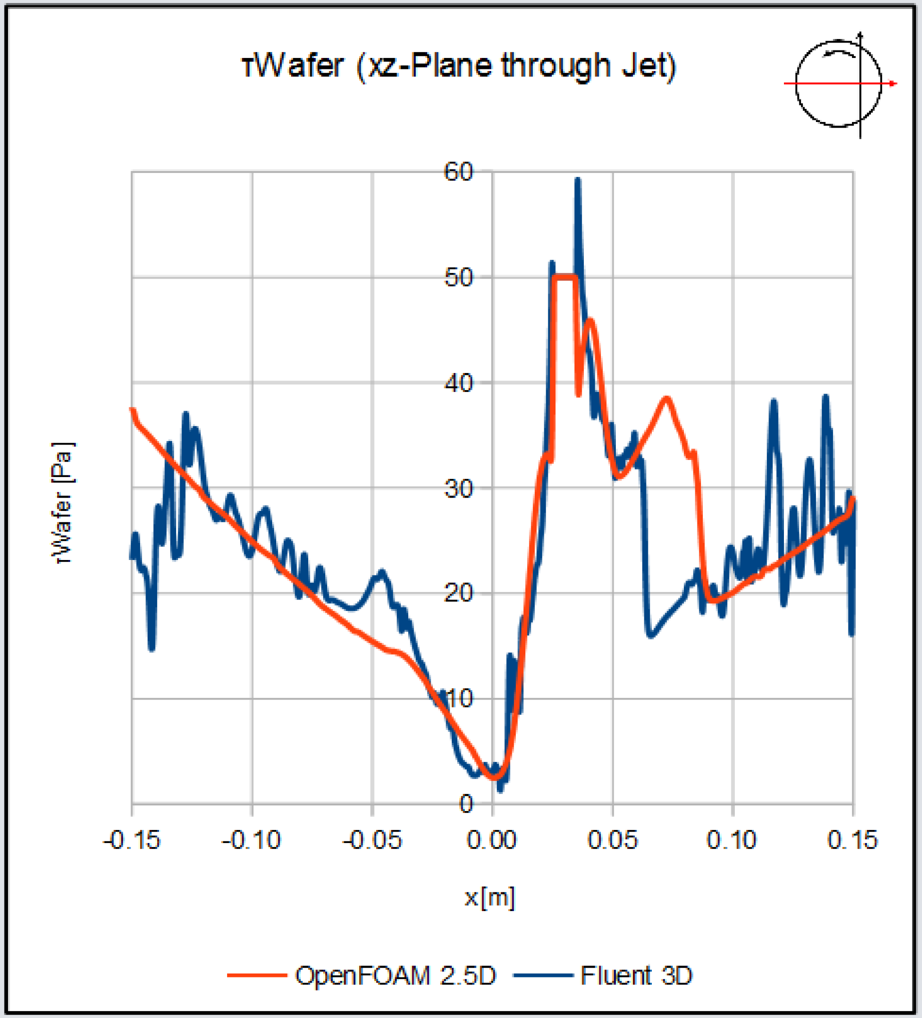


Case 1a: 500rpm, 1.5lpm, Δr=30mm, Spinetch-D



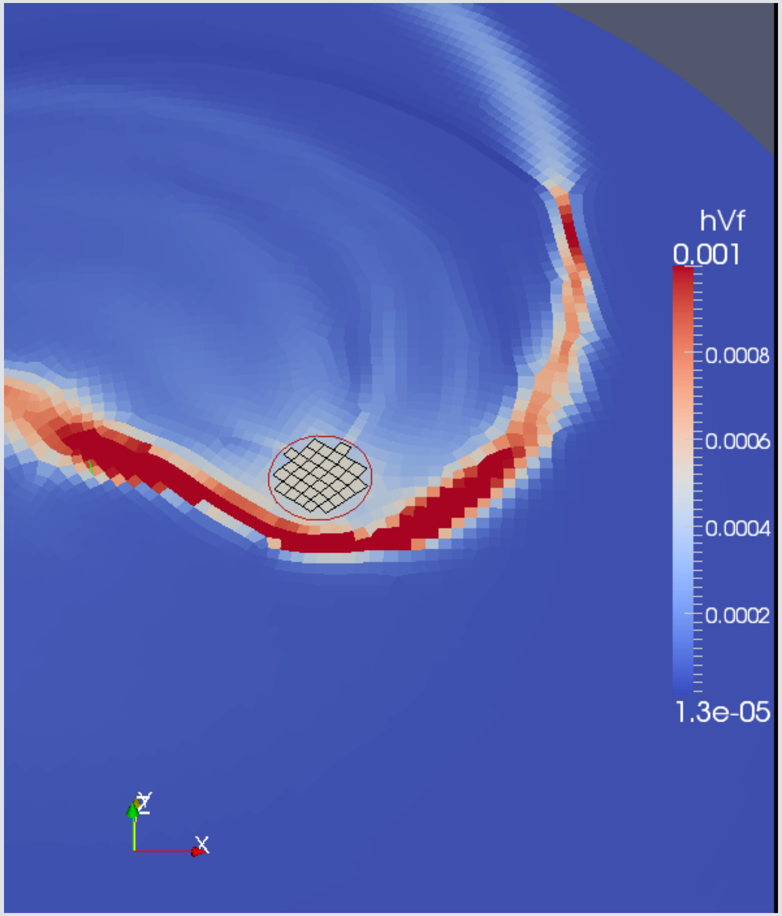
Case 1a: 500rpm, 1.5lpm, Δr=30mm, Spinetch-D



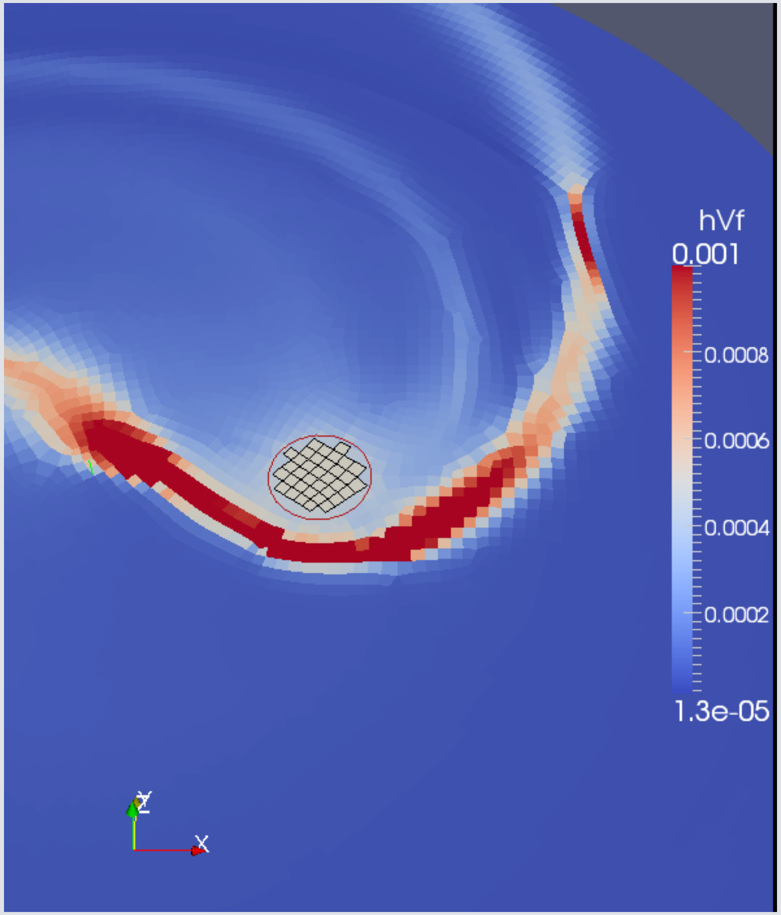


Impinging Jet: “Crown-Cap” Effect

Uncorrected Flow



Corrected Flow



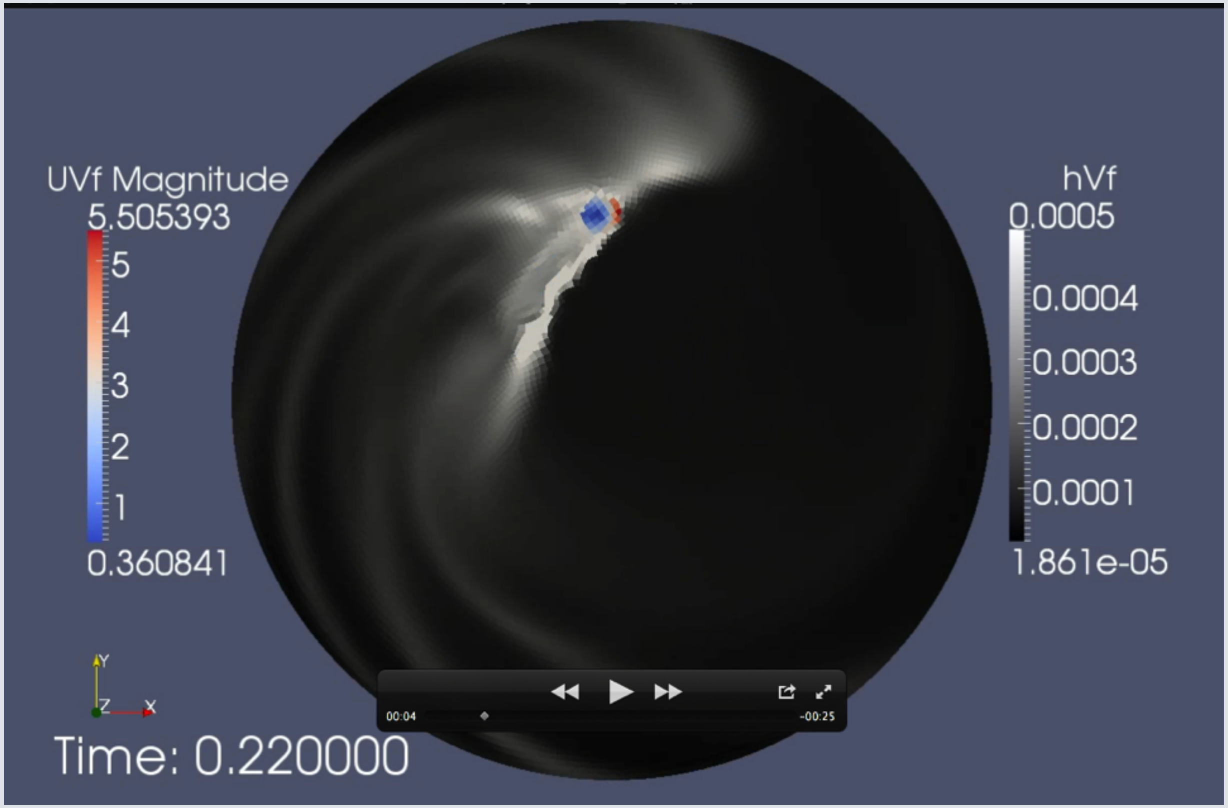
Animation of a moving inlet

Strömungsforschung GmbH

ICE

Innovative Computational Engineering

- **Black to white: height of the liquid film**
- **Color: prescribed velocity on the inlet**



Conclusion

- **2.5D solution shows a good agreement with 3D solution, while significantly saving on resources**
  - Solution in an impingement area has to be prescribed
  - Zone close to jet, influenced by the impingement, is showing a reasonable agreement and is still able to capture important effects
    - We never promised to be exact here!
  - Zone outside of the impingement influence is showing a very good agreement
  - Smooth solution without waviness
  - Small meshes and significantly shorter simulation times

Discussion

- Thank you for your attention!
- Questions?