An open source non-Newtonian FSI solver implementation

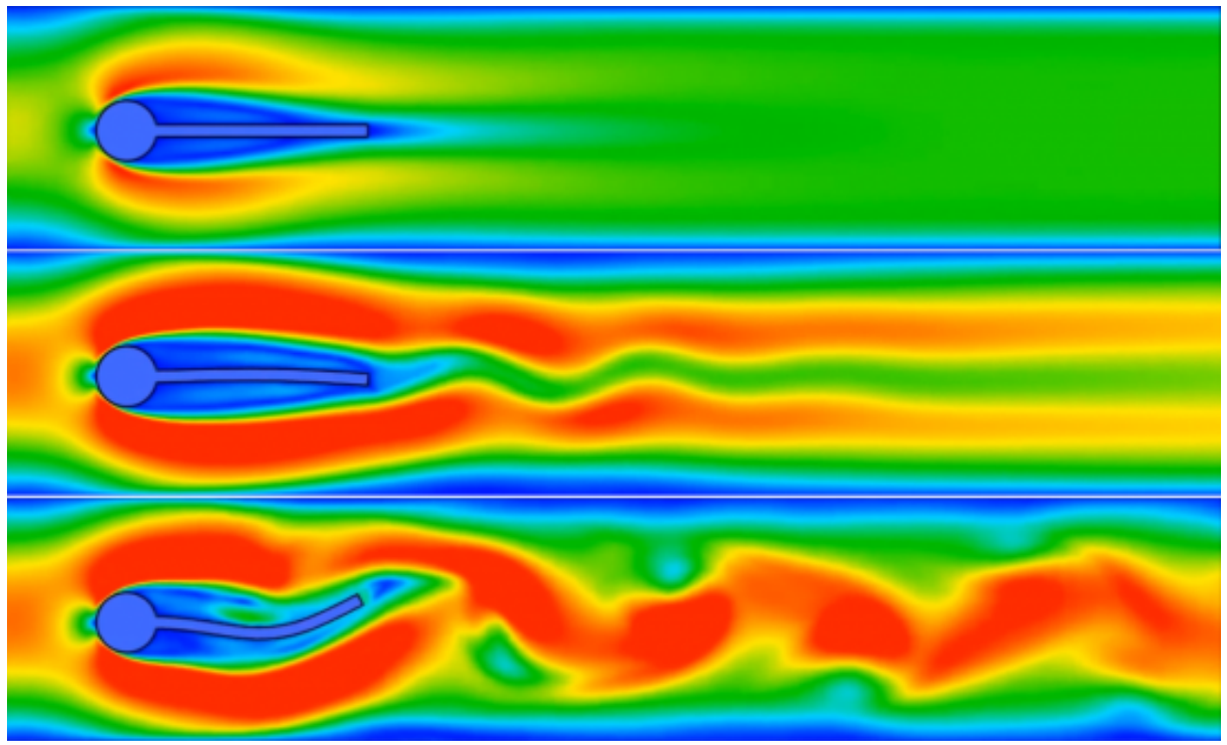
Multiphysics applications at GSC-CE and Numerical Analysis and Mathematical Modeling at FNB

Naples - Italy - May 25, 2017

Lucas Ribeiro de Azevedo

TECHNISCHE UNIVERSITÄT DARMSTADT

Supervisors:
Prof. Dr. rer. nat. Michael Schäfer
Prof. Dr. rer. nat. Dieter Bothe
PhD Francisco J. Galindo Rosales



Source: Varinex

Content

TECHNISCHE UNIVERSITÄT DARMSTADT

- Motivation
- Implementation
- Verification
- Continuous model
- Numerical model
- Results
- Conclusion
- Next steps
- Acknowledgements
- References

Motivation

TECHNISCHE UNIVERSITÄT DARMSTADT

- **A tool**: This work is part of a bigger research project
- **Goal**: Create a non-Newtonian Fluid Structure Interaction (nNFSI) solver
- **Means**: Based on an existing Newtonian FSI (NFSI) solver taking in consideration the principle of reusability of software
- **Challenges**: Lacking of a benchmarking papers and difficult numerical convergence
- **Related work**: Mostly with focus on NFSI solver
- **Contribution**: The whole scientific community can now use an open-source nNFSI solver that is free to use, distribute and modify

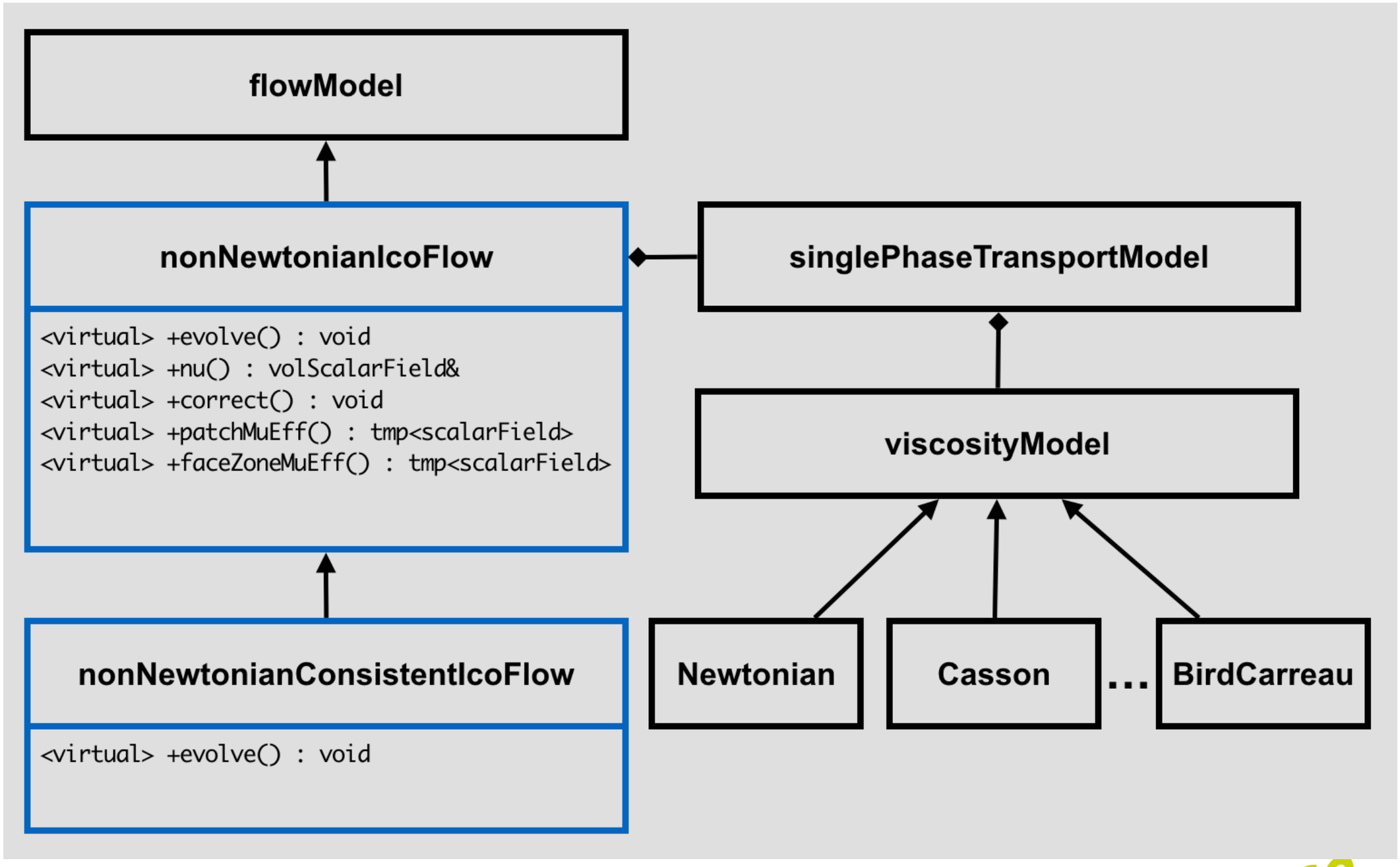
Implementation

TECHNISCHE UNIVERSITÄT DARMSTADT

- The base code is the FSI-toolkit developed for foam-extended-3.1 [Tukovic - 2014]
- **Shift of paradigm**: The FSI treats the flow and the structure solvers as classes that constitute the FSI library and not as executables, as usually happens in OpenFOAM.
- Two new classes were created
  - nonNewtonianIcoFlow
  - nonNewtonianConsistentIcoFlow

Implementation

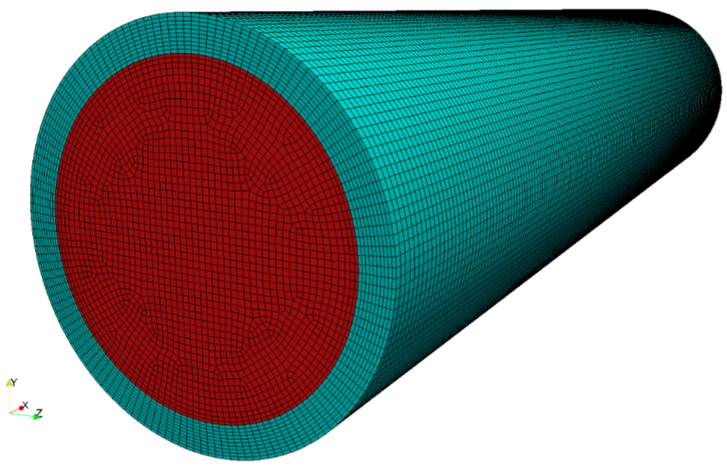
TECHNISCHE UNIVERSITÄT DARMSTADT



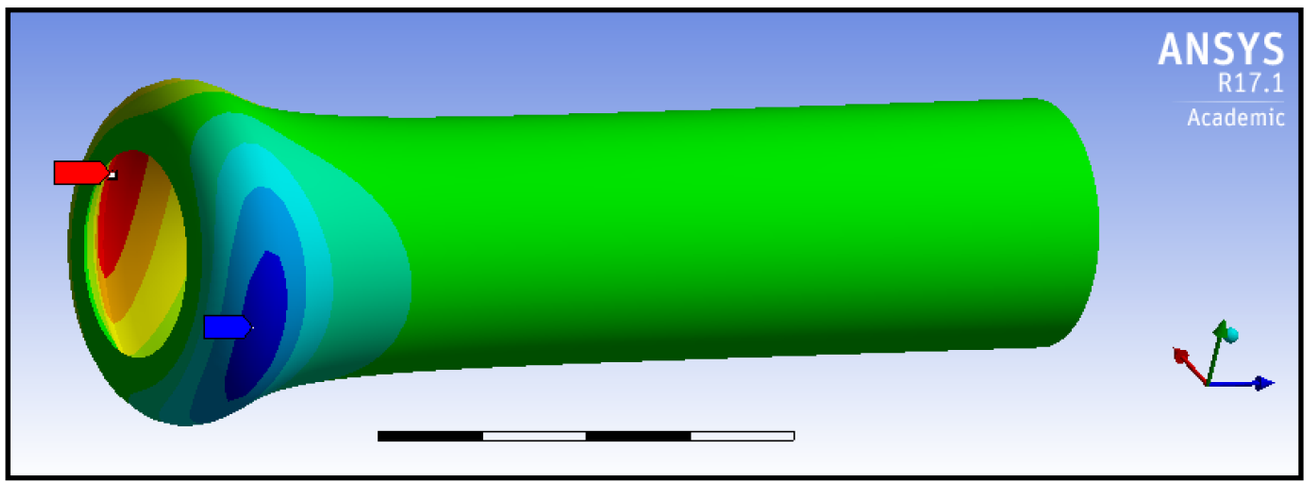
Verification

TECHNISCHE UNIVERSITÄT DARMSTADT

- The NFSI(**base solver**) was verified with well known benchmarks FSI [Tukovic - 2014]
- Simulation
- Classical pulse-in-vessel case, but with non-Newtonian fluid
- Very unstable
- During the first 0.003s, a uniform overpressure of 1333.2 N/m2 is applied at the inlet.
- The same simulation run both using OpenFOAM and ANSYS



a) The mesh



b) Deformed domain

Continuous model

TECHNISCHE UNIVERSITÄT DARMSTADT

- Fluid Model:
  - Transient and Isothermal
  - Shape of the domain is time-dependent ⇒ ALE ⇒ GCL
  - Incompressible and non-Newtonian
  - Based on mass and momentums conservation equations

- Solid Model:
  - Transient and Isothermal
  - Hyperelastic material
  - Based on momentums conservation equations

ALE: Arbitrary Lagrangian-Eulerian [Erwin Stein - 2004]
GCL: Geometrical Conservation Law [Demirdzic - 1987]

Numerical model

TECHNISCHE UNIVERSITÄT DARMSTADT

- Fluid and Solid models
  - FVM - Unstructured grid
  - Fluid mesh deformation governed by Laplacian mesh motion equation
  - PISO and GAMG for fluid
  - PCG for solid
  - Colocated grid for fluid using Rhie-Chow
  - Backward time integration (2nd order and three time levels)

FVM: Finite Volume Method
PISO: Pressure Implicit with Splitting of Operator
GAMG: Generalised geometric-Algebraic Multi-Grid
PCG: Preconditioned Conjugate Gradient

Numerical model

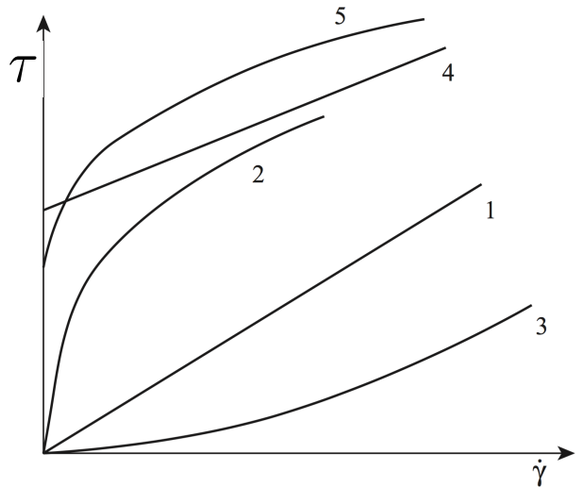
TECHNISCHE UNIVERSITÄT DARMSTADT

- FSI model
  - Partitioned, Strongly and Implicit coupling
  - Decomposed using Dirichlet-Neuman
  - Fluid solver uses displacement of the FSI
  - Solid solver uses pressure force on the interface
  - Coupling scheme is the IQN-ILS

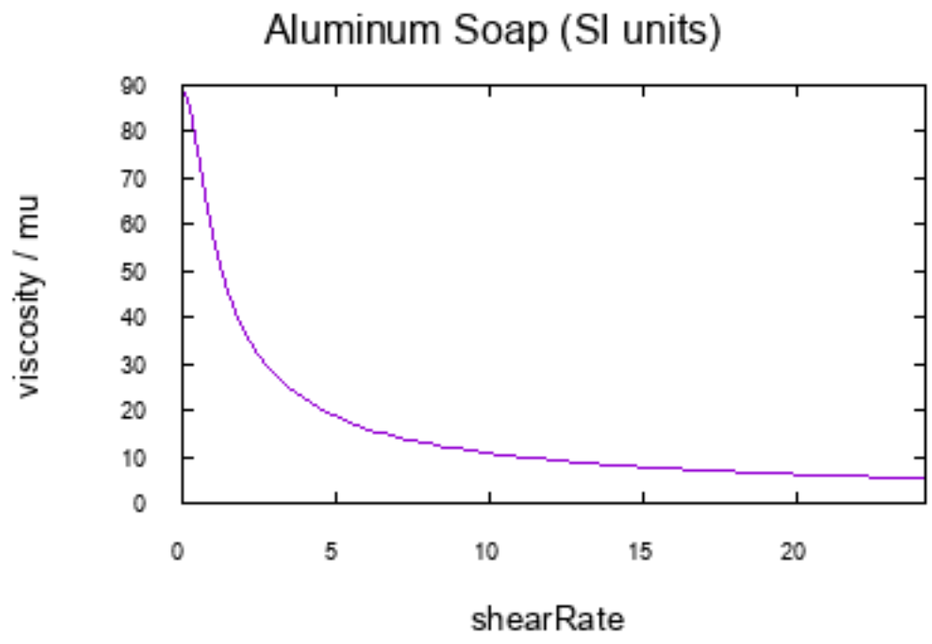
IQN-ILS: interface quasi-Newton technique with an approximation for the inverse of the Jacobian from a least-squares model [Degroote - 2010]

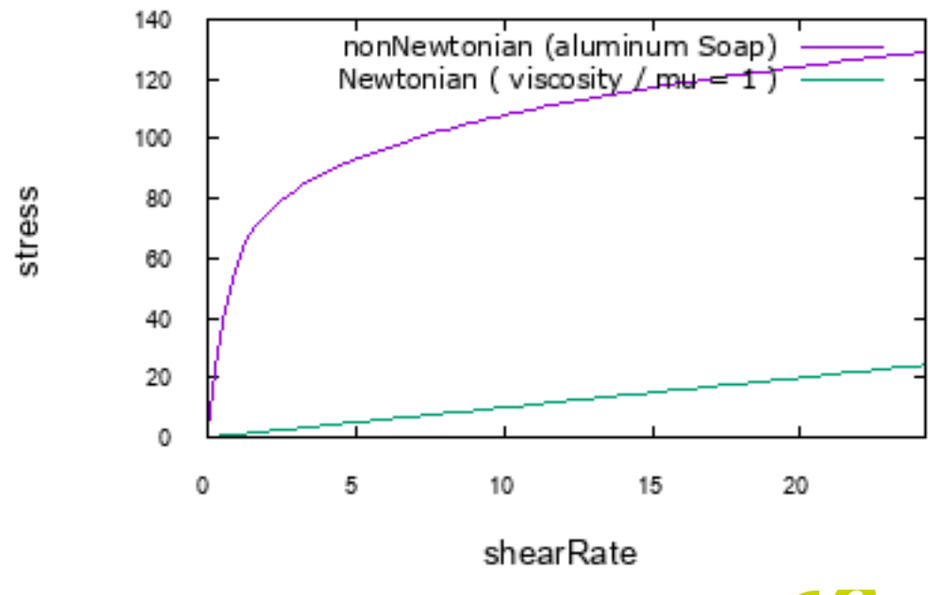
Results

$$\tau = \mu(\dot{\gamma})\dot{\gamma}$$

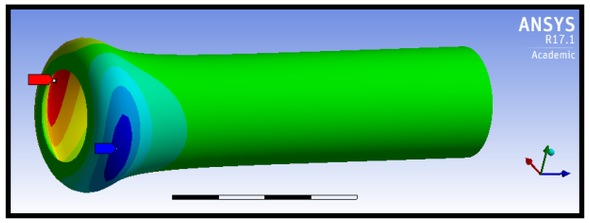


1 - Newtonian : Water
2 - Shear thinning : Silicone coatings
3 - Shear thickening : Corn starch in water
4 - Bingham plastic : Toothpaste
5 - Bingham pseudoplastic : Paints





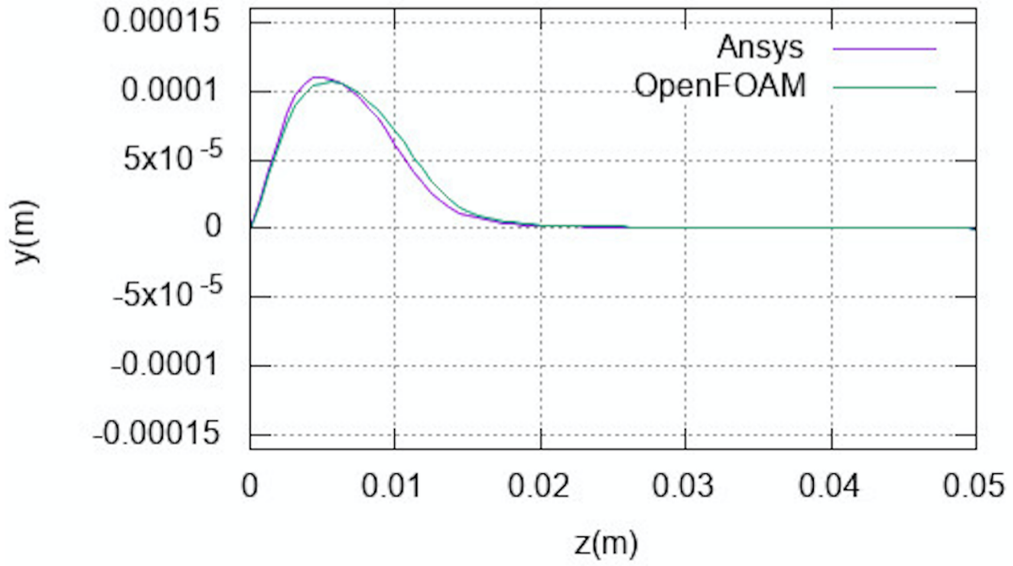
Results



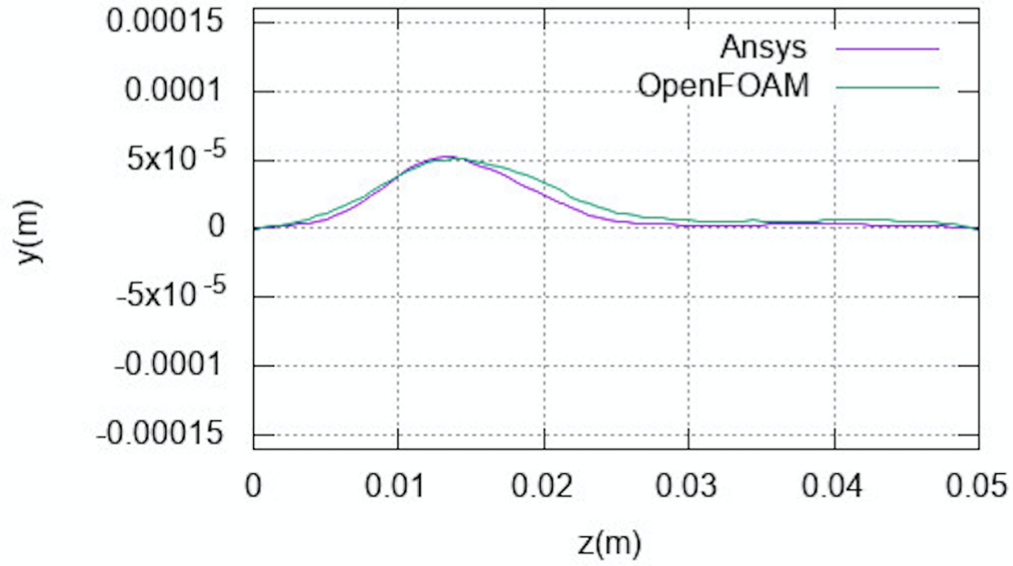
TECHNISCHE UNIVERSITÄT DARMSTADT

Aluminum soap and Hyperelastic

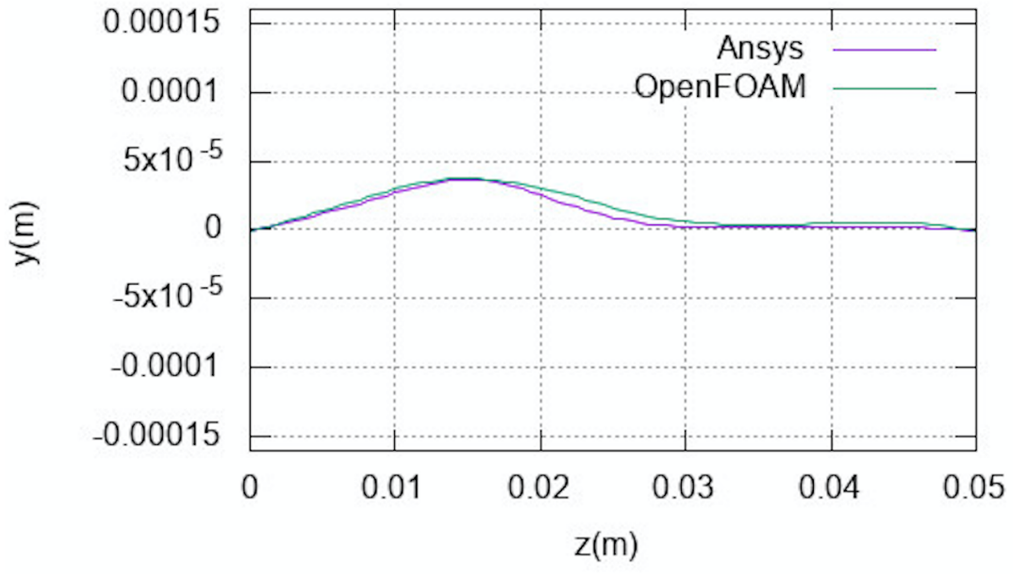
Time = 0.003s



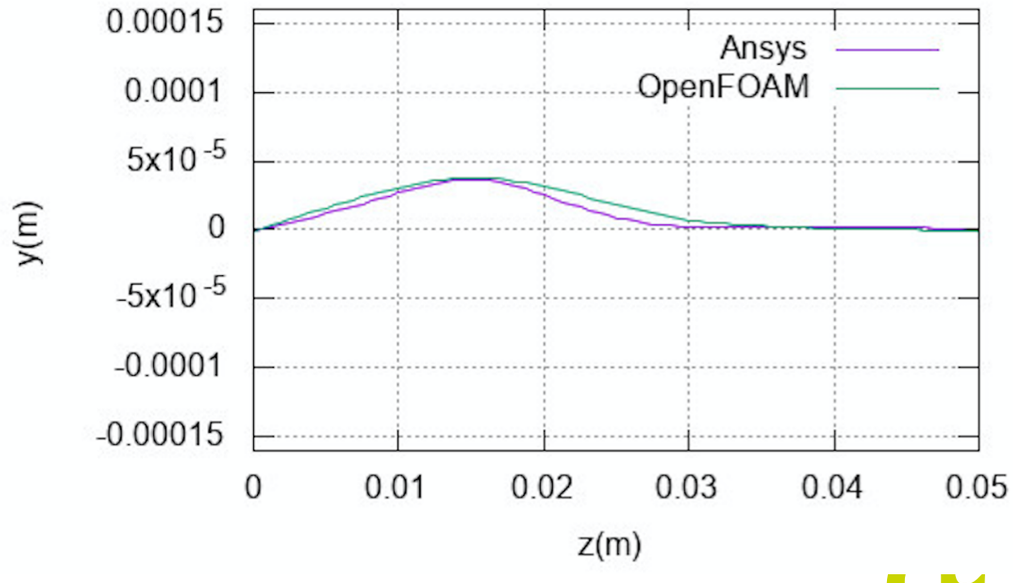
Time = 0.006s



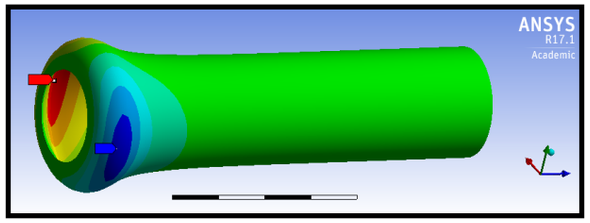
Time = 0.009s



Time = 0.012s



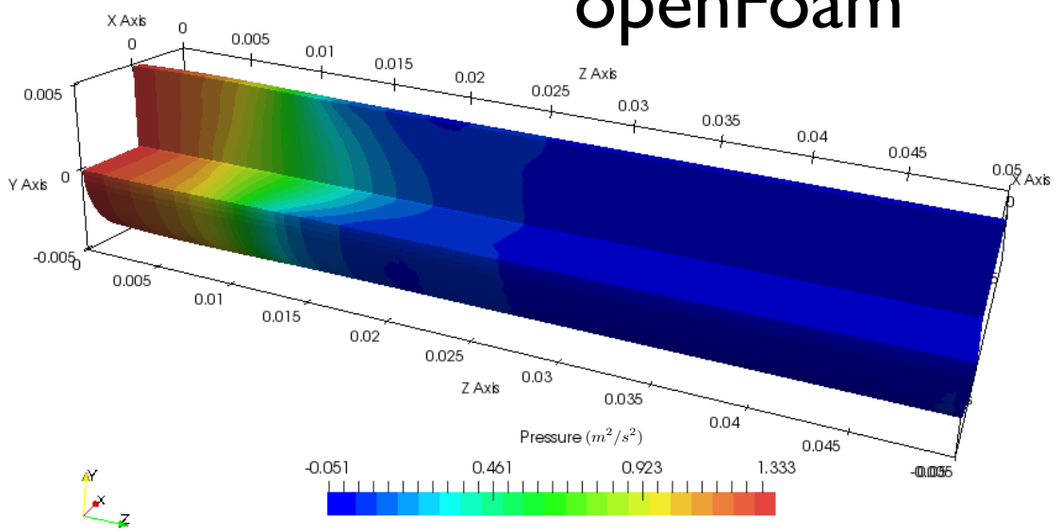
Results



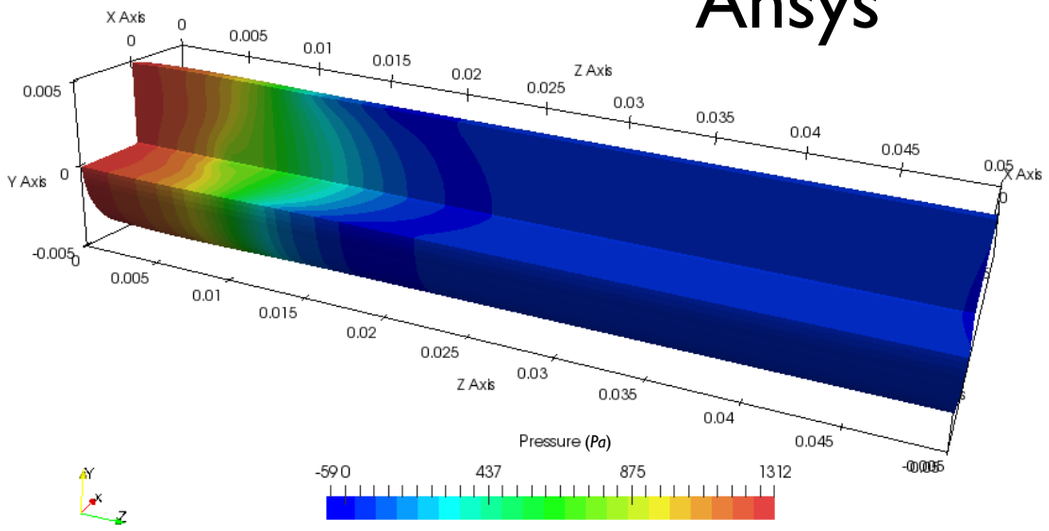
TECHNISCHE UNIVERSITÄT DARMSTADT

Pressure field

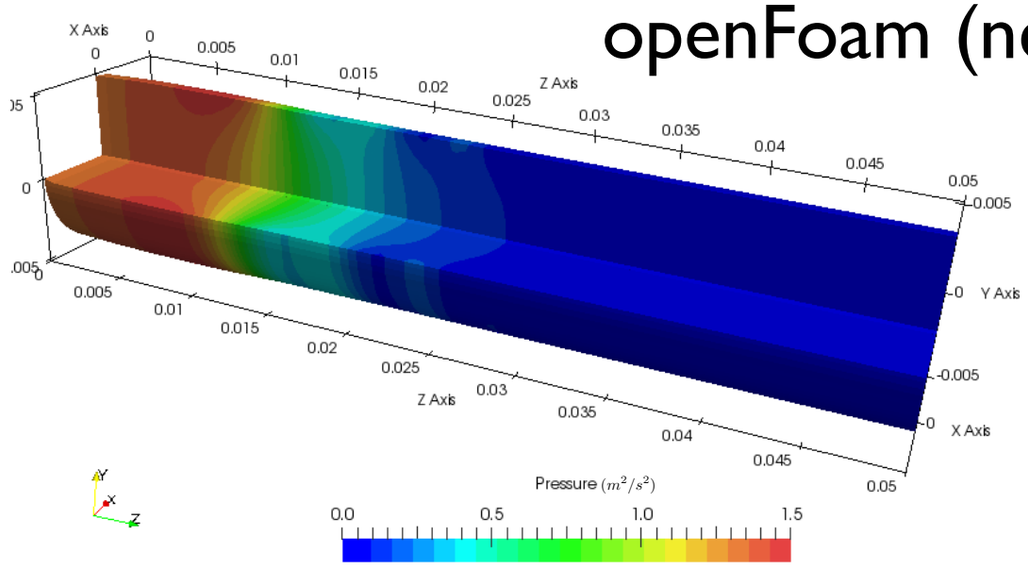
openFoam



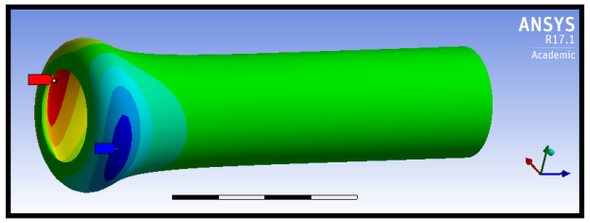
Ansys



openFoam (newtonian)



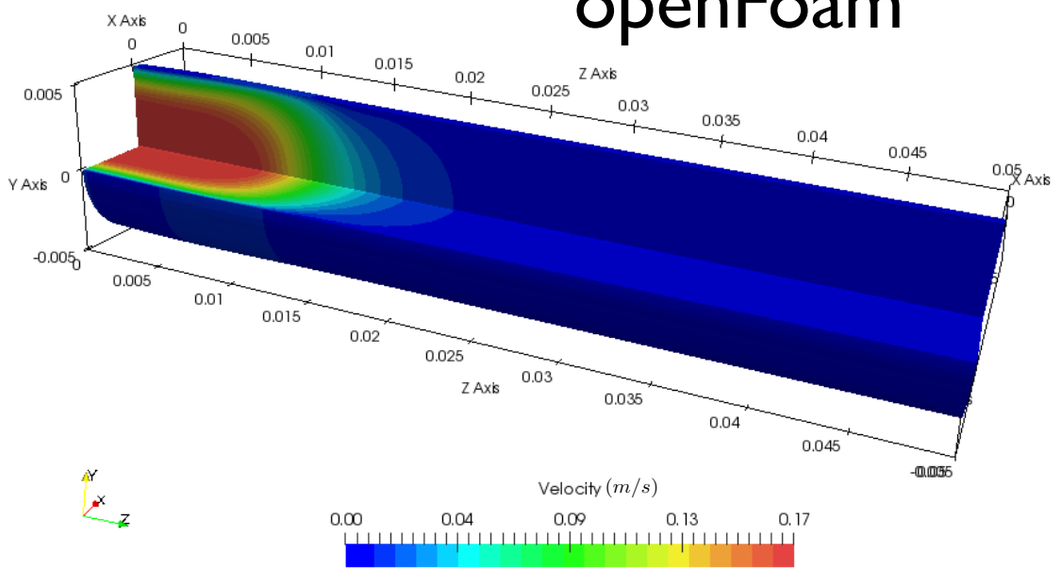
Results



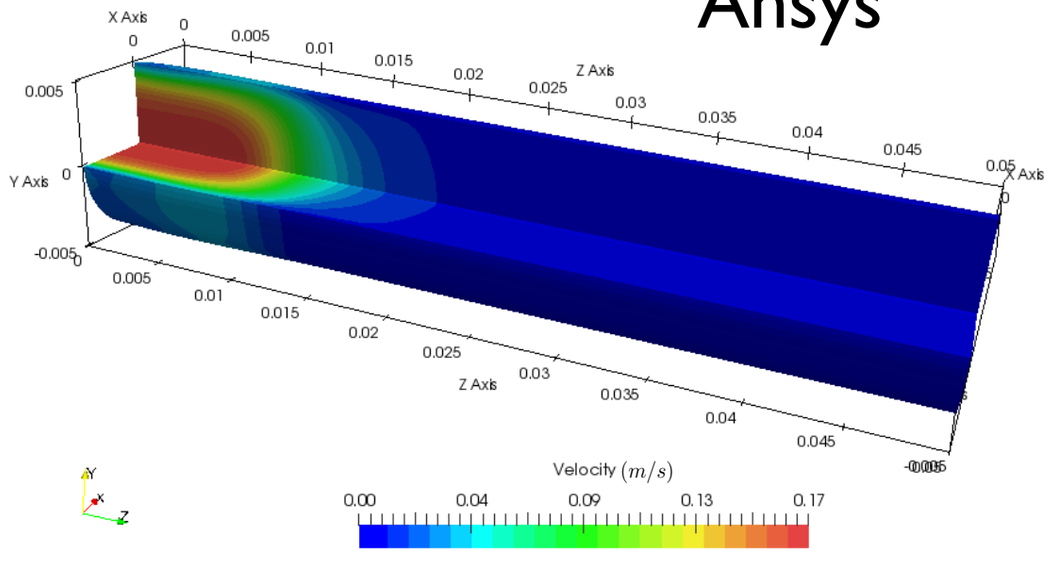
TECHNISCHE UNIVERSITÄT DARMSTADT

Axial velocity field

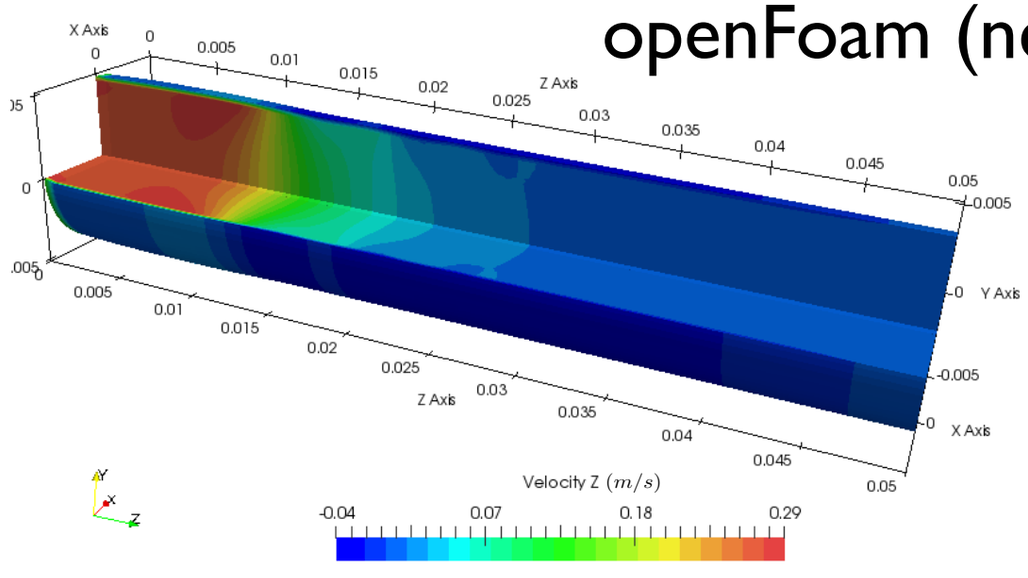
openFoam



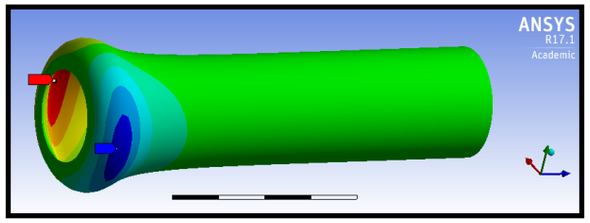
Ansys



openFoam (newtonian)



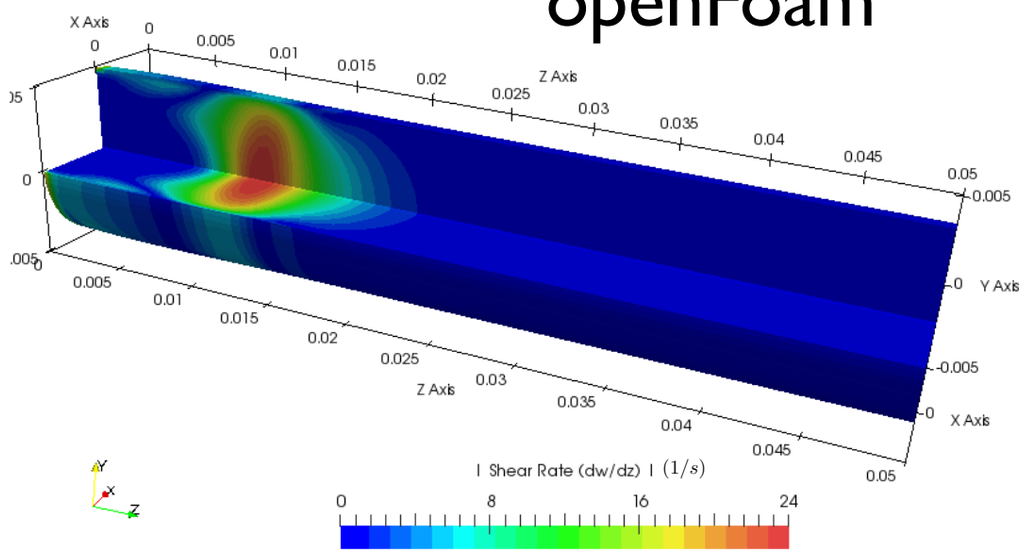
Results



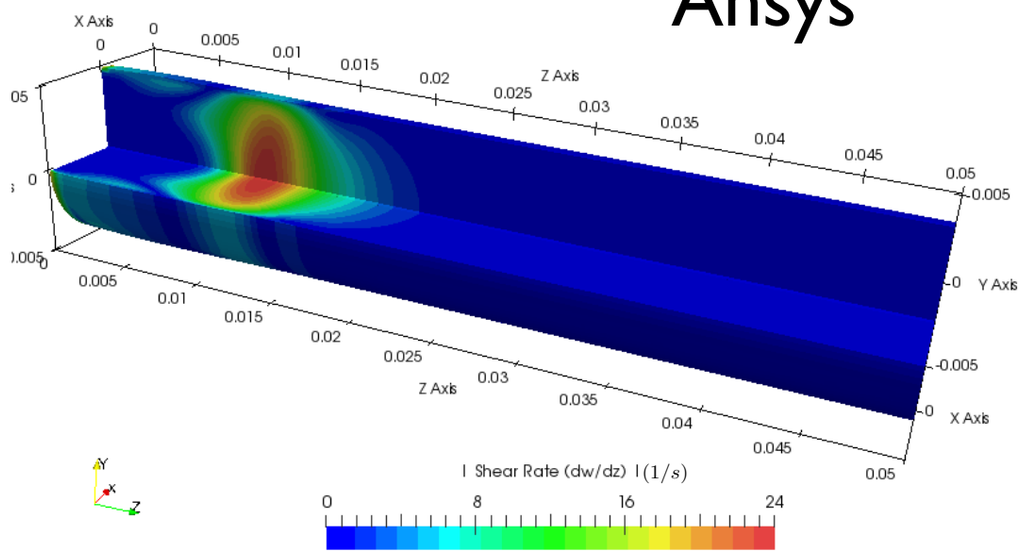
TECHNISCHE UNIVERSITÄT DARMSTADT

Shear Rate

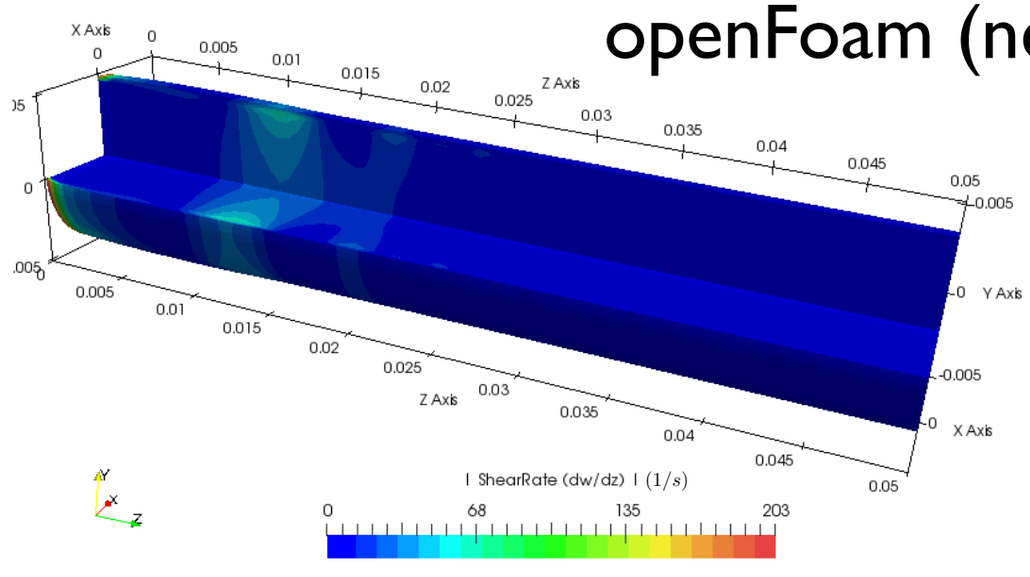
openFoam



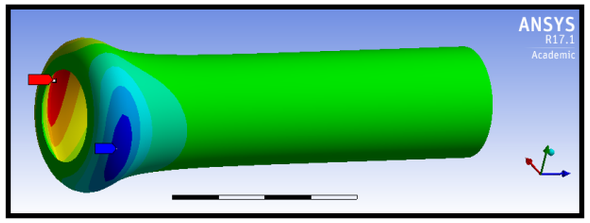
Ansys



openFoam (newtonian)



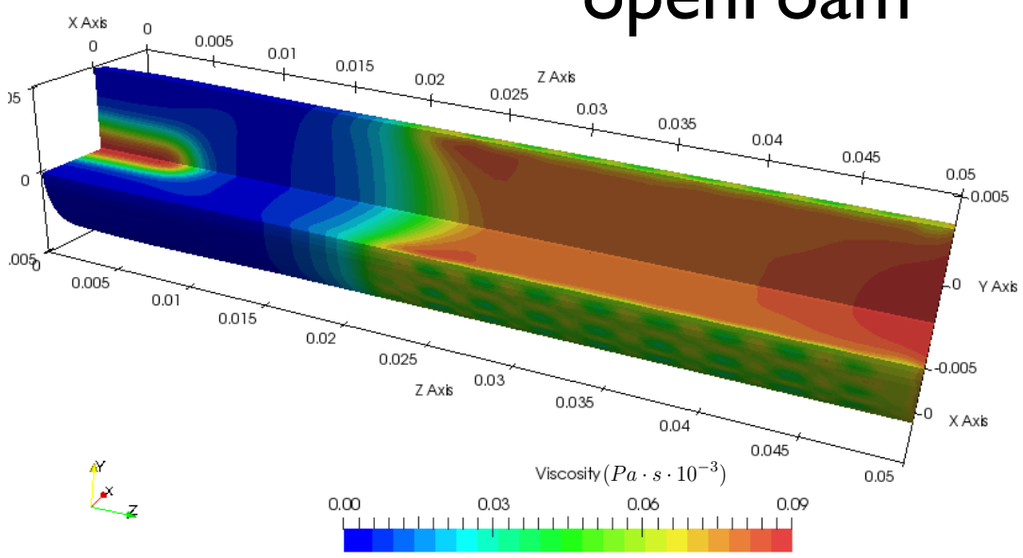
Results



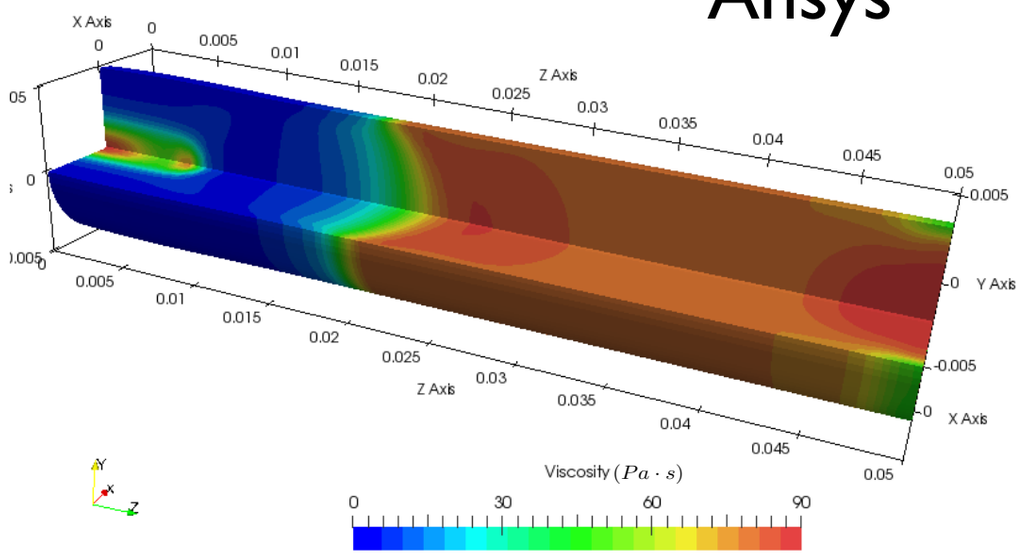
TECHNISCHE UNIVERSITÄT DARMSTADT

Viscosity

openFoam



Ansys



Conclusion

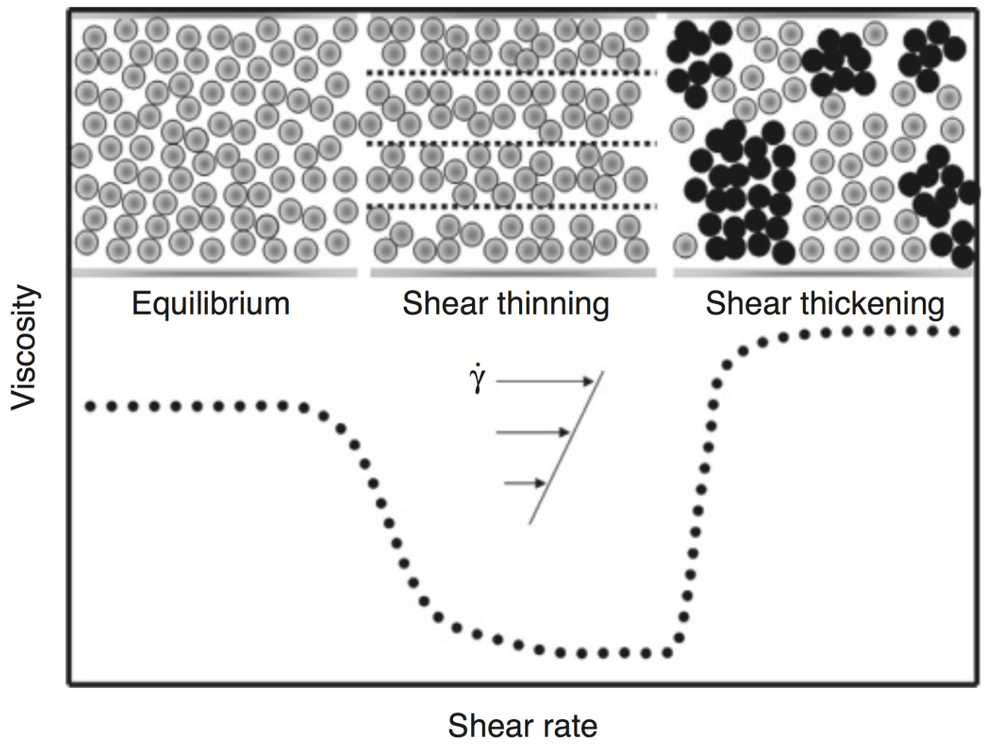
TECHNISCHE UNIVERSITÄT DARMSTADT

- General purpose nNFSI solver
- “Unified” approach
- Objected-oriented approach
- All old non-Newtonian models can be used
- Easy to extend to new models
- Solver is verified

Next steps

TECHNISCHE UNIVERSITÄT DARMSTADT

- Implement the Ogden hyperelastic model
- Implement a viscoelastic fluid model
- Implement a shear thickening fluid model



Acknowledgements

This work is supported by the 'Excellence Initiative' of the German Federal and State Governments and the Graduate School of Computational Engineering at Technische Universität Darmstadt

Thank you!

References

- [Fridtjov Irgens - 2014] Rheology and Non-Newtonian Fluids
- [Wagner - 2009] Shear thickening in colloidal dispersions
- [Degroote - 2010] Development of Algorithms for the Partitioned Simulation of Strongly Coupled Fluid-Structure Interaction Problems
- [Demirdzic - 1987] Space conservation law in finite volume calculations of fluid flow
- [Erwin Stein - 2004] Encyclopedia of computational mechanics - Fluids - Volume 1
- [Tukovic - 2014] Z. Tukovic, P. Cardiff, A. Karac, H. Jasak and A. Ivankovic: Parallel unstructured finite-volume method for fluid-structure interaction, manuscript in preparation