# DEVELOPMENT OF A COUPLED INCOMPRESSIBLE VISCOELASTIC FLUID FLOW SOLVER BASED ON FOAM-EXTEND

C. FERNANDES[1], V. VUKČEVIĆ[2], T. UROIĆ[2], O.S. CARNEIRO[1], H. JASAK[2,3] AND J.M. NÓBREGA[1]

[1]*Institute for Polymers and Composites/i3N, University of Minho, Campus de Azurém, 4800-058 Guimarães, Portugal, cbpf@dep.uminho.pt*

[2]*Faculty of Mechanical Engineering and Naval Architecture, Ivana Lučića 5, 10000 Zagreb, Croatia*

[3]*Wikki Ltd, 459 Southbank House, SE1 7SJ, London, United Kingdom*

***Keywords:*** *coupled pressure-velocity-extra stress tensor, block solver, foam-extend*

To help designing polymer processing tools it is essential to use efficient computational fluid dynamics numerical codes, which can save time and resources when compared to those usually spent on experimental trial-and-error processes. The design tasks are even more difficult to undertake, both experimentally and numerically, when dealing with complex rheology materials, as happens with the viscoelastic fluids, which present some counter intuitive phenomena. On the computational side, for this type of fluids a non-linear constitutive equation that includes elastic effects must be considered. In this framework, for an isothermal problem, the set of highly coupled governing equations, consisting of the continuity, the momentum and the constitutive equations, has to be solved.

The usual approaches employed are based on segregated algorithms, in which all the equations are solved iteratively and sequentially [1]. Due to the strong coupling between the governing equations, this sequential iterative process is quite sensitive and prone to divergence. Additionally, the segregated approach is known to present slow convergence (due to the requirement of using high relaxation factors), and limit substantially the Deborah number (a measure of the elasticity relevance) that can be achieved in the numerical calculation.

This work describes a new numerical code, in the context of the finite-volume method, which follows a coupled approach to compute the flow of viscoelastic fluids. The code was implemented in the open-source computational library foam-extend [2], a community driven fork of the OpenFOAM® software. The solution of the enlarged system of equations, composed by continuity, momentum and extra-stress constitutive equations, is obtained using an algebraic multigrid solver. The performance of the coupled viscoelastic solver is assessed with two case studies, namely the Oldroyd-B Poiseuille and the UCM lid-driven cavity flows, targeting to evaluate the advantages in terms of the number of iterations and CPU time, with increasing mesh density and Deborah number. Additionally, for verification purposes, the results obtained with the developed code are compared with analytical solutions and results obtained with the segregated version of the viscoelastic modelling code [1].

**Acknowledgements**

This work is funded by UID/CTM/50025/2013 - LA0025, with the financial support of FCT/MEC through national funds and when applicable by FEDER co-funded, within the partnership agreement PT2020. The authors would like to acknowledge the Minho University cluster under the project Search-ON2: Revitalization of HPC infrastructure of UMinho, (NORTE-07-0162-FEDER-000086), co-funded by the North Portugal Regional Operational Programme (ON.2-0 Novo Norte), under the National Strategic Reference Framework (NSRF), through the European Regional Development Fund (ERDF). The authors thank the COST ACTION MP1305 – Flowing Matter for the financial support given to the visit of C. Fernandes to the University of Zagreb.

**References**

[1] C. Fernandes, M. S. B. Araujo, L. L. Ferrás, and J. M. Nóbrega. Improved Both Sides Diffusion (iBSD): a new and straightforward stabilization approach for viscoelastic fluids flows. J. of Non-Newtonian Fluid Mech., 249:63-78, 2017.

[2] foam-extend. Open Source CFD Toolbox, 2013. URL https://sourceforge.net/projects/foam-extend/