[1] Ngamaramvaranggul, V., Webster, M.F.; *Int. J. Numer. Meth. Fluids* **2002**, *38*, 677-710.

Tuesday 4:25 Room 207 CR12

A numerical investigation of flow-type sensitive fluids

Flavia Zinani[1] and Sergio L. Frey[2]
[1]Mechanical Engineering, Universidade do Vale do Rio dos Sinos UNISINOS, São Leopoldo, RS 93022-000, Brazil; [2]Department of Mechanical Engineering, Federal University of Rio Grande do Sul, Porto Alegre, Rio Grande do Sul 90050-070, Brazil

This paper introduces stabilized finite element approximations for flow-type sensitive fluid flows. A quasi-Newtonian model, based on a kinematic parameter of flow classification and shear and extensional viscosities, is employed to represent the fluid behavior from pure shear to pure extension.Their major features are the viscosity function dependent of the flow type and the extra-stress tensor described by the generalized Newtonian model. The quasi-Newtonian fluid flows herein considered are approximated by a multi-field Galerkin least-squares (GLS) method in terms of strain rate, pressure and velocity. Due to the addition of residual-based least-squares terms of flow governing equations, the GLS method allows the use of simple combinations of finite element interpolations and remains stable even in flows subjected to high geometric and material non-linearity. The flow domain is a four-to-one abrupt planar contraction and the triple (D-p-u) is approximated by a combination of bi-linear Lagrangian interpolation for pressure, and bi-quadratic ones for the strain rate and velocity. The bi-quadratic interpolation for the tensor D assures an accurate representation of the flow classifier that depends on the first derivatives of D. For a relevant range of the Deborah number for such a problem (De from 0 to 0.6), three flow-type sensitive fluids are investigated: (i) a shear-thinning fluid; (ii) an extension-thickening fluid; and (iii) a shear-thinning and extension-thickening fluid. For all fluids, the distribution of the flow classifier is evaluated, capturing both extensional flow regions, in the contraction vicinity, and pure shear flows, far away from the contraction. The preliminary numerical approximations proved to be sufficiently stable and physically meaningful, encouraging in this way to deepen the numerical research of such a model.

Tuesday 4:50 Room 207 CR13

Computational modeling of high Deborah number flow and elastic instability in planar contraction geometry

Youngdon Kwon
School of Chemical Engineering, Sungkyunkwan University, Suwon, Republic of Korea

Especially in highly nonlinear regime, elastic fluid such as polymeric liquid displays quite unique and various flow behaviors, drastically different from Newtonian one and often against our common intuition. Experiments, theoretical stability analyses and direct computational studies have been conducted to accumulate knowledge on this area of industrial as well as scientific significance. Computational modeling of these unique elastic flow phenomena, especially the purely elastic flow instabilities has been a challenging topic in rheology, which is far from complete resolution even after 50 years' elaborate research conducted by numerous scientists. In this work, we present and discuss the current status of understanding this problem recently achieved by the current author especially for the unstable contraction flow in 2D creeping flow asymptotics. In modeling we employ the essence of the mathematical stability results analyzed for constitutive equations, and then develop and implement appropriate computational algorithm for robust discrete continuation and fast time integration of governing equations. The algorithm demonstrates stable computation in highly nonlinear flow regime and the result exhibits symmetry breaking and oscillation of corner vortices in 4:1 contraction flow, which may be understood as purely elastic flow instability.

Tuesday 5:15 Room 207 CR14

Implementation of the wall slip boundary condition in a computational rheology code based on the finite volume method

Luis L. Ferrás[1], João M. Nóbrega[1], and Fernando T. Pinho[2]
[1]Institute for Polymers and Composites, University of Minho, Guimarães, Portugal; [2]CEFT, Departamento de Engenharia Mecânica, Universidade do Porto, Porto 4200-465, Portugal

Although the no-slip boundary condition applies to most flows, there is experimental evidence for the existence of wall slip in many flows. Hence, for the sake of modeling accuracy it is important for computational methods to be able to consider this particular condition.

Most works on the implementation of the wall slip boundary condition concern Finite Element Method based approaches, while for the Finite Volume Method (FVM) the information is scarce. To fill this gap, we present two different numerical techniques for the implementation of linear and non-linear Navier slip boundary conditions, into a computational rheology code based on the FVM. The implementation is initially developed for channel flow and subsequently extended to geometries requiring non-orthogonal structured meshes and also to non-Newtonian fluid. A comparison between these new methods and the usual iterative process (update the boundary condition at the end of each iteration) is also presented, showing that for this new approach the convergence issues during the iterative procedure are solved.

With the proposed methodology and for the linear slip law, the implementation at the boundaries is fully implicit and convergence does not require relaxation, even when the slip coefficient is high. For non-linear slip laws, two alternative approaches for the numerical solution of the governing equations are also investigated: one requiring the solution of a transcendental equation at the boundary, and the other based on the linearization of the slip-stress vector relationship. The employment of the wall slip is illustrated with the numerical investigation of the flow in a 4:1 planar contraction of Newtonian and viscoelastic fluids described by the linear simplified Phan-Thien-Tanner rheological equation. While for Newtonian fluids the vortex size reduces with the increase of the slip level, for the viscoelastic fluid the size of the separated flow region remains approximately unchanged, but its shape is affected.