Simulation of generalized Newtonian fluids with the Smoothed Particle Hydrodynamics method

Miloš Simeunović1,* and Holger Steeb1

1 Institute of Applied Mechanics (CE), University Stuttgart, Pfaffenwaldring 7, 70569 Stuttgart, Germany

In this paper we examine Direct Numerical Simulation DNS of single phase non-Newtonian fluids described by a generalized Newtonian rheology. Non-Newtonian flow including large deformations and free surface flow could be observed in a wide range of industrial and environmental applications. For that reason, we choose the Lagrangian Smoothed Particle Hydrodynamics (SPH) as a simulation tool. The non-Newtonian solver is implemented into the general purpose particle framework HOOMD-blue [1,2], which allows for massive parallel CPU and GPU simulations. Numerical accuracy of the model is demonstrated by simulating confined Poiseuille flow between parallel plates. The validation has been done for a wide variety of shear strain–shear stress ratios where also the effect of the solid-fluid interface boundary condition is examined. Furthermore, a comparison with the experimental results for the broken dam problem [3] showed good accuracy of the free-surface flow prediction. Additionally, we have investigated a slump test (Abrams cone [4]), a relatively simple but in practice often performed experiment for non-Newtonian yield-stress fluids. Finally, we have investigated in further detail the cone diameter, height, flow time, the final shape and the influence of the rheological model parameters as well as the boundary conditions related to the stationary cone configuration.

1 Introduction and motivation

In our work we perform Direct numerical simulation by using weakly compressible numerical environment for the meshfree Smoothed particle hydrodynamics method that was originally developed in 1977 by pioneering works of Monaghan and Gingold [6]. Like other particle mesh-less methods, SPH is especially attractive for solving problems including large deformations and free surface flow. In SPH every integration point stores problem related variables such as density, pressure or viscosity. The main idea is to provide accurate and stable numerical computations in a domain discretized by set of arbitrarily distributed particles.

2 Smoothed particle hydrodynamics and balance equations

The basic principle of the SPH method is the approximation a field variable \( \Phi(x, t) \) using a set of disordered integration points also interpreted as particles. The concept of the integral representation of \( \Phi(x, t) \) defined over the computational domain \( \Omega \) has the following mathematical form of a convolution integral

\[
\Phi(x) = \int_\Omega \Phi(x') W(x - x', h) \, dv
\]

where \( W \) is a so-called interpolation kernel characterized by a smoothing length \( h \) defining the domain of influence of \( W \) itself. We consider governing equations of the transient, viscous, isothermal and quasi incompressible fluid flow in Lagrangian framework. With this set of assumptions, mass and momentum conservation can be respectively written

\[
\dot{\rho} = -\rho \nabla \cdot \mathbf{v}, \quad \text{and} \quad \rho \dot{\mathbf{v}} = -\nabla p + \rho \mathbf{g}
\]

where \( t \) is time, \( \mathbf{v} \) is velocity vector, \( \mathbf{g} \) is the volumetric force, \( p \) is hydrostatic pressure and \( \tau \) is deviatoric part of the stress tensor. Assuming weak compressibility of the fluid, the pressure \( p \) is computed by using an equation of state as given in [5].

For fluids exhibiting a non-Newtonian rheological properties, the viscous term in the governing equation has to be treated separately. Assuming that dynamical viscosity of the generalized Newtonian fluid \( \mu \) depends on the second principal invariant \( I_2 \) of the shear strain rate \( \mathbf{D} \) and by introducing constitutive law we can write

\[
\tau = \mu(I_2)\mathbf{D}.
\]

In the context of a SPH discretization, the mass and momentum balance as also boundary conditions treatment can be expressed in a different forms and for the numerical implementation we followed [5].

* Corresponding author: e-mail milos.simeunovic@mechbau.uni-stuttgart.de, phone +49 (0)711 685 60876, fax +49 (0)711 685 66347

This is an open access article under the terms of the Creative Commons Attribution License 4.0, which permits use, distribution and reproduction in any medium, provided the original work is properly cited.
3 Model validation

Validation has been performed using a simple uni directional flow between the parallel plates driven by the body force also known as Poiseuille flow. For the constitutive model we used the truncated power law model where the velocity profile in fully developed flow regime has analytical solution according to [7]. The validation results (Fig. 1) showed good matching to the analytical solution for the wide range of the flow index $n$ parameter, representing both - shear thinning and thickening behavior.

![Fig. 1: Comparison between SPH simulation results (marked with red dots) and the analytical solution (marked by blue line) for the flow between parallel plates at the distance $H$ for different cases of power law exponent $n$.](image)

4 Application problems

We have studied classical 3-dimensional benchmark problems often used to test the quality of SPH implementations. In both cases we investigate problems including free surfaces. The motion is driven by the body force and we observe the evolution of the characteristic regions and the final distribution formed when the steady state has been reached.

![Fig. 2: Evolution of the leading front of the dam-break problem at different times $t = 0.18, 0.3, 1.0, 1.8s$.](image)

![Fig. 3: Slump test and the evolution of the dynamic viscosity.](image)

5 Conclusion and outlook

In this work we have implemented a method for solving non-Newtonian free surface flow. A high performance numerical environment framework has been used and the comparison to the analytical solution of Poiseuille problem has shown perfect results. Our further intention is to investigate the material properties and parameters and identify them through an inverse numerical parameter identification process with the assistance of numerical simulation.

Acknowledgements  The authors gratefully acknowledge funding from the German Research Foundation (DFG) for the research priority program SPP 2005 “Opus Fluidum Futurum - Rheology of reactive, multiscale, multiphase construction materials”.

References

[1] Anderson, J. A.; Lorenz, C. D. & Travesset, A. General purpose molecular dynamics simulations fully implemented on graphics processing units. Journal of Computational Physics, 2008, 227, 5342-5359.

[2] Glaser, J.; Nguyen, T. D.; Anderson, J. A.; Lui, P.; Spiga, F.; Millan, J. A.; Morse, D. C. & Glotzer, S. C. Strong scaling of general-purpose molecular dynamics simulations on GPUs. Computer Physics Communications, 2015, 192, 91-107.

[3] Komatina, D. & Jovanovic, M. Experimental study of steady and unsteady free surface flows with water-clay mixtures. Journal of Hydraulic Research, 1997, 35, 579-590.

[4] Roussel, N. Correlation between yield stress and slump: Comparison between numerical simulations and concrete rheometers results. Materials and Structures, 2006, 39, 501-509.

[5] Adami S., Hu X., & Adams N. A generalized wall boundary condition for smoothed particle hydrodynamics. Journal Computational Physics, 231(21):7057–7075, 2012.

[6] Gingold R. A.& Monaghan J. J. Smoothed particle hydrodynamics - theory and application to non-spherical stars. Mon. Not. R. Astron. Soc., 181:375–389, 1977.

[7] R. Bird. Dynamics of Polymeric Liquids. v. Wiley, 1977.