

Radka Keslerová; David Trdlička; Hynek Řezníček

Numerical modelling of steady and unsteady flows of generalized Newtonian fluids

In: Jan Chleboun and Pavel Kůs and Petr Příkryl and Karel Segeth and Jakub Šístek and Tomáš Vejchodský (eds.):  
Programs and Algorithms of Numerical Mathematics, Proceedings of Seminar. Janov nad Nisou, June 19-24, 2016.  
Institute of Mathematics CAS, Prague, 2017. pp. 55–62.

Persistent URL: <http://dml.cz/dmlcz/702998>

**Terms of use:**

© Institute of Mathematics CAS, 2017

Institute of Mathematics of the Czech Academy of Sciences provides access to digitized documents strictly for personal use. Each copy of any part of this document must contain these *Terms of use*.



This document has been digitized, optimized for electronic delivery and stamped with digital signature within the project *DML-CZ: The Czech Digital Mathematics Library*  
<http://dml.cz>

## NUMERICAL MODELLING OF STEADY AND UNSTEADY FLOWS OF GENERALIZED NEWTONIAN FLUIDS

Radka Keslerová, David Trdlička, Hynek Řezníček

Czech Technical University in Prague, Faculty of Mechanical Engineering,  
Department of Technical Mathematics,  
Karlovo nám. 13, 121 35 Prague, Czech Republic

Radka.Keslerova@fs.cvut.cz, David.Trdlicka@fs.cvut.cz, Hynek.Reznicek@fs.cvut.cz

**Abstract:** This work presents the numerical solution of laminar incompressible viscous flow in a three dimensional branching channel with circular cross section for generalized Newtonian fluids. This model can be generalized by cross model in shear thinning meaning. The governing system of equations is based on the system of balance laws for mass and momentum. Numerical tests are performed on a three dimensional geometry, the branching channel with one entrance and two outlet parts. Numerical solution of the described model is based on central finite volume method using explicit Runge–Kutta time integration. The steady state solution is achieved for  $t \rightarrow \infty$ . In this case the artificial compressibility method will be applied. In the case of unsteady computation artificial compressibility method is considered.

**Keywords:** viscous fluids flow, generalized Newtonian fluids, cross model, finite volume method, Runge-Kutta scheme, artificial compressibility method

**MSC:** 65L06, 65N08, 76A05, 76A10, 76D05

### 1. Introduction

Branching of pipes occurs in many technical or biological applications. In [13] the effects of viscoelasticity on the pitchfork bifurcation using a numerical finite volume method was investigated. Results from both the upper-convected Maxwell and Oldroyd-B models show that the instability occurs at lower Reynolds numbers for viscoelastic fluids in comparison to the Newtonian base case. In [12] computational fluid dynamics simulations of steady viscoelastic flows through a planar two dimensional T-junction is considered and the influence of constitutive model and fluid elasticity upon the main recirculating flow characteristics formed at the junction and the shear stress fields is studied. In [4] a comparative numerical study of non-Newtonian fluid models capturing shear-thinning and viscoelastic effects of blood flow in idealized and realistic stenosed vessels was presented. Reference [2] proposes to analyze the

pulsatile flow of blood through catheterized stenosed artery considering the Carreau-Yasuda model. The effects of tapering angle, body acceleration and magnetic field are also taken into account. In [1] the unsteady pulsatile magneto-hydrodynamic blood flows through porous arteries concerning the influence of externally imposed periodic body acceleration and a periodic pressure gradient are numerically simulated. Paper [7] shows the mathematical models that represent non-Newtonian flow of blood through a stenosed artery in the presence of a transverse magnetic field. Here, the rheology of the flowing blood is characterised by a generalised Power law model.

In previous works [9] and [10] we studied the numerical simulation of generalized Newtonian and Oldroyd-B fluids flow in 2D branching channel. In this article the problem of the unsteady numerical simulation for the generalized Newtonian fluids flow is presented. The modelled domain is the three dimensional branching channel with T-junction.

## 2. Mathematical model

The governing system of equations is the system of generalized Navier-Stokes equations, see [3]. This system consists of the continuity equation

$$\operatorname{div} \mathbf{u} = 0 \quad (1)$$

and the momentum equation

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla P + \operatorname{div} \mathbf{T}, \quad (2)$$

where  $P$  is the pressure,  $\rho$  is the constant density,  $\mathbf{u}$  is the velocity vector. The symbol  $\mathbf{T}$  represents the stress tensor.

For the viscous fluids Newtonian mathematical model for the definition of the stress tensor  $\mathbf{T}$  is considered (see e.g. [4])

$$\mathbf{T} = 2\mu \mathbf{D}, \quad (3)$$

where  $\mu$  is the dynamic viscosity and tensor  $\mathbf{D}$  is the symmetric part of the velocity gradient,  $\mathbf{D} = \frac{1}{2}(\nabla \mathbf{u} + \nabla \mathbf{u}^T)$ .

For the numerical modelling of the generalized Newtonian fluids flow it is necessary to generalize the mathematical models. In this case the viscosity function  $\mu\dot{\gamma}$  is defined by cross model (for more details see [8, 14, 15])

$$\mu(\dot{\gamma}) = \mu_{\infty} + \frac{\mu_0 - \mu_{\infty}}{(1 + (\lambda\dot{\gamma})^b)^a}, \quad \dot{\gamma} = 2\sqrt{\frac{1}{2}\operatorname{tr} \mathbf{D}^2}, \quad (4)$$

with dynamical viscosities  $\mu_0 = 1.6 \cdot 10^{-1} Pa \cdot s$ ,  $\mu_{\infty} = 3.6 \cdot 10^{-3} Pa \cdot s$ , constants  $a = 1.23, b = 0.64$  and time parameter  $\lambda = 8.2 s$ . For Newtonian flow modelling, the viscosity is kept constant and equal to  $\mu_{\infty}$ . The relationships between dynamic viscosity  $\mu$  and shear rate  $\dot{\gamma}$  is shown in Fig. 1.

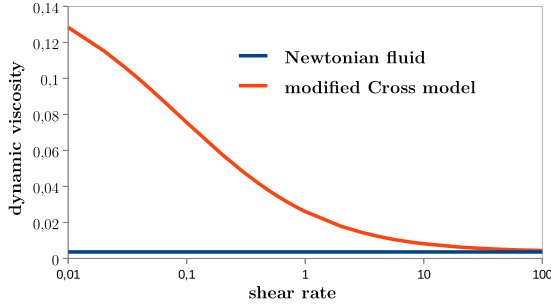


Figure 1: Viscosity  $\mu$  - shear rate  $\dot{\gamma}$  relationships for the two selected viscosity models.

### 3. Numerical solution

The mathematical models described above are solved numerically by the artificial compressibility approach combined with the finite-volume discretization. The artificial compressibility method [5, 6, 8, 9] is used to obtain an equation for pressure. It means that the continuity equation is completed by a pressure time derivative term  $\frac{\partial p}{\beta^2 \partial t}$ , where  $\beta$  is a positive parameter, making the inviscid part of the system of equations hyperbolic

$$\frac{1}{\beta^2} \frac{\partial p}{\partial t} + \operatorname{div} \mathbf{u} = 0. \quad (5)$$

The parameter  $\beta$  for steady simulation is chosen equal to the maximum inlet velocity. This value ensures good convergence to steady state but is not large enough to make the transient solution accurate in time. Therefore it is suitable for steady flows only. The system including the modified continuity equation and the momentum equations can be written

$$\tilde{R}_\beta W_t + F_x^c + G_y^c + H_z^c = F_x^v + G_y^v + H_z^v, \quad \tilde{R}_\beta = \operatorname{diag}\left(\frac{1}{\beta^2}, 1, \dots, 1\right), \quad (6)$$

where  $W$  is the vector of unknowns. By superscripts  $c$  and  $v$  we denote the inviscid and the viscous fluxes, respectively.

The space discretization is done by a cell-centered finite-volume method with hexahedral finite volumes [6, 8, 11]. The arising system of ODEs is integrated in time by the explicit multistage Runge–Kutta scheme [9].

The flow is modelled in a bounded computational domain where a boundary is divided into three mutually disjoint parts: a solid wall, an outlet and an inlet. At the inlet Dirichlet boundary condition for velocity vector and for the stress tensor is used. For the pressure, homogeneous Neumann boundary condition is used. At the outlet parts the pressure value is prescribed and for the velocity vector and the stress tensor homogeneous Neumann boundary condition is used. The no-slip boundary condition for the velocity vector is used on the wall. For the pressure and stress tensor homogeneous Neumann boundary condition is considered.

### 3.1. Unsteady computation

For numerical solution of unsteady flows the artificial compressibility method with unsteady boundary conditions is applied [6]. This method consists in modifying continuity equation in the same way as was described above. The question arises, how big the artificial compressibility parameter  $\beta$  should be in numerical simulation. Numerical calculations show that  $\beta = 6 \text{ m} \cdot \text{s}^{-1}$  (100x higher than for steady simulation) is the proper choice.

Two different forms of unsteady boundary conditions are considered. First, prescribing the pressure value at the outlet (branch). Second, prescribing the normal velocity component  $u$  at the inlet as

$$up = up_{in} (1 + K \sin(\omega t)). \quad (7)$$

Here  $\omega$  is the angular velocity defined as  $\omega = 2\pi f$ , where  $f$  is a frequency. The symbol  $up$  denotes the pressure or the velocity according to used unsteady boundary condition. Symbol  $up_{in}$  is the constant value obtained from the steady simulation. Constant  $K$  is for the pressure 0.5 and for the velocity it is equal to reference velocity value. Other boundary condition are considered as in the steady case.

In this work three values of the frequency are tested:  $f = 2, 5, 10 \text{ Hz}$ .

## 4. Numerical results

This section deals with the comparison of the numerical results of generalized Newtonian fluids flow for steady case. Numerical tests are performed in an idealized branching channel with the circular cross-section. Fig. 2 (left) shows the shape of the tested domain. The computational domain is discretized using a block-structured, wall fitted mesh with hexahedral cells. The domain is divided to 19 blocks with 125 000 cells.

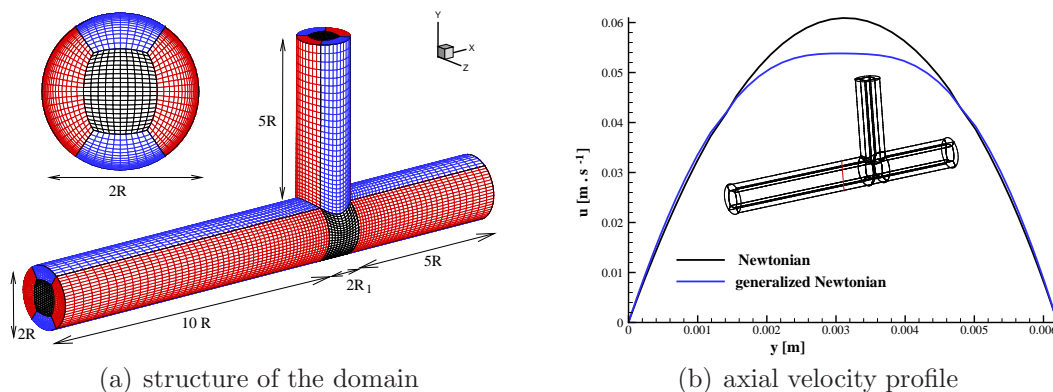


Figure 2: Structure of the tested domain (left) and axial velocity profile of tested fluids (right). Reference radius  $R = 0.0031 \text{ m}$ .

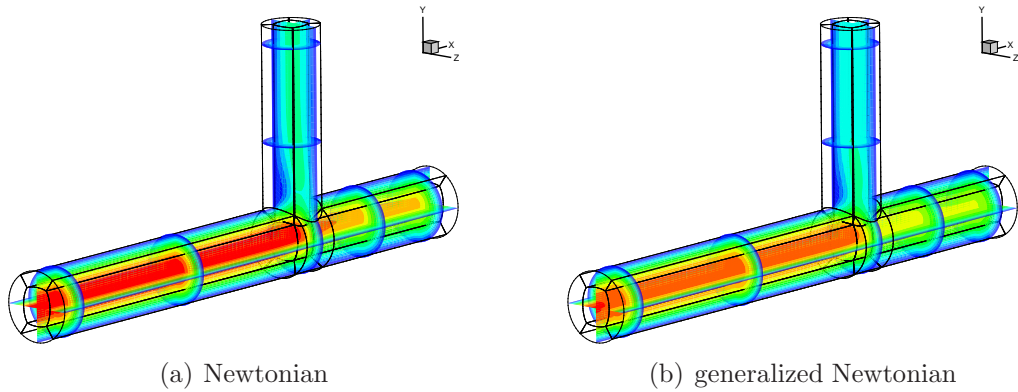


Figure 3: Velocity isolines of steady flows for generalized Newtonian fluids.

As initial condition the following model parameters are used: reference radius  $R = 0.0031 \text{ m}$ , branch radius  $R_1 = 0.0025 \text{ m}$ , dynamic viscosity  $\mu = 0.0036 \text{ Pa} \cdot \text{s}$ , reference velocity  $U_0 = 0.0615 \text{ m} \cdot \text{s}^{-1}$ , constant density  $\rho = 1050 \text{ kg} \cdot \text{m}^{-3}$ . At the inlet the Dirichlet boundary conditions for velocity are used, the parabolic profile with reference velocity value  $U_0$ . At the outlet the constant pressure values are prescribed. In Fig. 2 the axial velocity profile close to the branching is shown. The line for Newtonian fluids is similar to the parabolic line, as was assumed. It is clear that the shear-thinning fluids attain lower maximum velocity in the central part of the channel (close to the axis of symmetry) which is compensated by the increase of local velocity in the boundary layer close to the wall.

In Fig. 3 the velocity isolines and the cuts through the channel are shown. The axial velocity isolines in the center-plane area for tested fluids are shown in Fig. 4. It can be observed from these that the size of separation region for generalized Newtonian fluids is smaller than for Newtonian fluids.

These steady numerical results are used as initial condition for unsteady numerical computation. The artificial compressibility method with high  $\beta$  parameter is used for the unsteady numerical simulation.

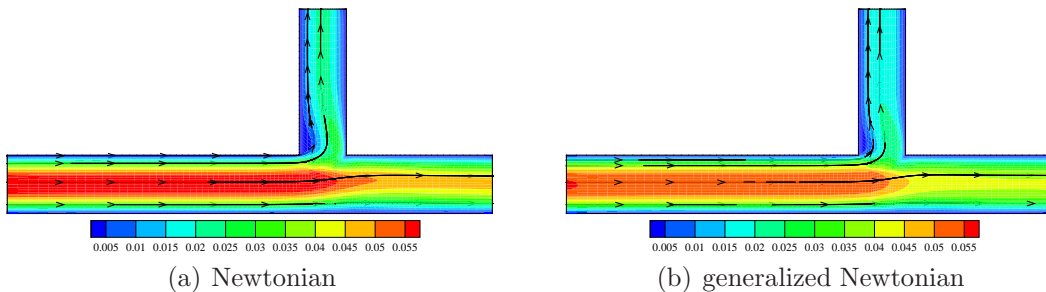


Figure 4: Axial velocity isolines in the center-plane area for generalized Newtonian fluids.

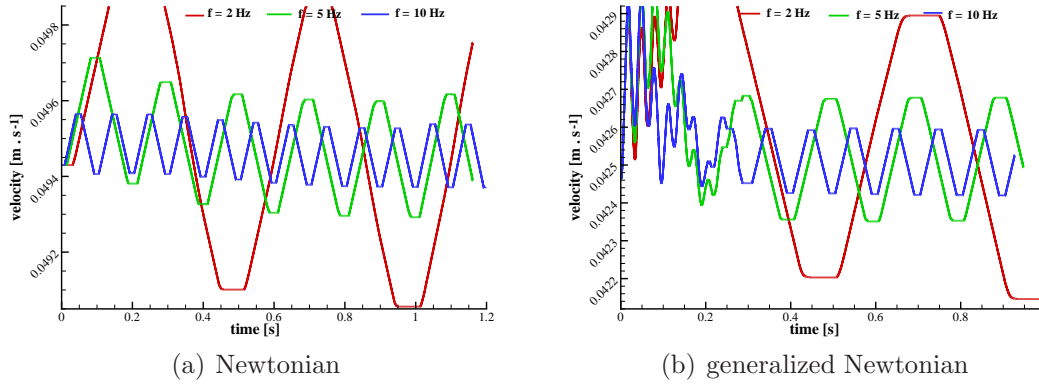


Figure 5: The graphs of the velocity as the function of time for three tested values of frequency - oscillation of the pressure at the outlet.

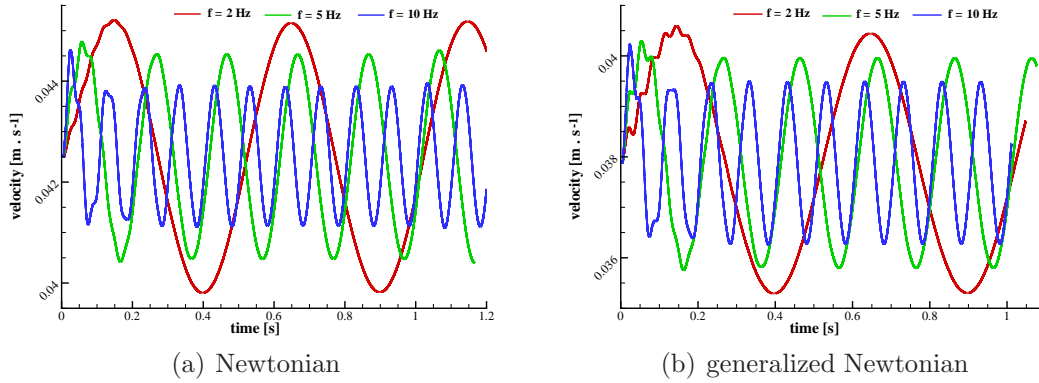


Figure 6: The graphs of the velocity as the function of time for three tested values of frequency - pulsation of the velocity at the inlet.

Figs. 5 and 6 show the magnitude of the velocity during the time period. The velocity is taken from the point inside the domain. In Fig. 5 the pressure at the outlet (branch) and in Fig. 6 the velocity at the inlet are prescribed by (7).

The numerical results (the magnitude of the velocity) are affected by the choice of the oscillation's variable (pressure or velocity) as well as by the velocity field. In the case where the boundary function (7) was used for pulsation of the velocity at the inlet, the numerical results (Fig. 6) for Newtonian and generalized Newtonian fluids are very similar independently of the position in the domain. On the other hand if the oscillation of the pressure at the outlet is considered than the resulting magnitudes of the velocity (taken in the same point as in the previous case) are different for Newtonian and generalised Newtonian fluids (6). It is affected by the type of fluid and by the behaviour of fluids in the branching (the size of separation region).

## 5. Conclusion

Classical Newtonian model as well as its generalized (shear-thinning) modification have been considered to model flow in the branching channel with T-junction. The goal of this study is to investigate shear-thinning effects in steady flow simulations. From the presented velocity profile, it is clear that the shear thinning fluids (generalized Newtonian fluids) attain lower maximum velocity in the central part of the channel (close to the axis of symmetry) which is compensated by the increase of local velocity in the boundary layer close to the wall.

The numerical method used to solve the governing equations seems to be sufficiently robust and efficient for the appropriate resolution of the given class of problems.

For unsteady simulation Newtonian and generalized Newtonian fluids were considered. An artificial compressibility approach was considered for numerical solution of unsteady governing equations. The artificial compressibility parameter  $\beta$  was set to be  $6 \text{ m} \cdot \text{s}^{-1}$ . Several values of frequency were tested. Pressure value in the branch outlet and the velocity value in the inlet were prescribed by a periodic function. The numerical results given by graphs of the velocity as the function of time in the point in the domain were presented.

For the future work, extending this unsteady simulation for generalized Oldroyd-B fluids flow will be considered. The dual-time stepping method will be used.

## Acknowledgments

This work was supported by the grant SGS16/206/OHK2/3T/12.

## References

- [1] Akbarzadeh, P.: Pulsatile magneto-hydrodynamic blood flows through porous blood vessels using a third grade non-Newtonian fluids model. *Computer Methods and Programs in Biomedicine* **126** (2016), 3–19.
- [2] Ali, N., Zaman, A., Sajid, M., Nietoc, J. J., and Torres, A.: Unsteady non-Newtonian blood flow through a tapered overlapping stenosed catheterized vessel. *Math. Biosci.* **269** (2015), 94–103.
- [3] Beneš, L., Louda, P., Kozel, K., Keslerová, R., and Štigler, J.: Numerical simulations of flow through channels with T-junction. *Appl. Math. Comput.* **219** (2013), 7225–7235.
- [4] Bodnar, T., Sequeira, A., and Prosi, M.: On the shear-thinning and viscoelastic effects of blood flow under various flow rates. *Appl. Math. Comput.* **217** (2010), 5055–5067.
- [5] Chorin, A. J.: A numerical method for solving incompressible viscous flow problem. *J. Comput. Phys.* **135** (1967), 118–125.



- [6] Honzátko, R.: *Numerical simulations of incompressible flows with dynamical and aeroelastic effects*. Ph.D. thesis, Czech Technical University in Prague, 2007.
- [7] Ikbāl, M. A., Chakravarty, S., Wong, K. K. L., Mazumdar, J., and Mandal, P. K.: Unsteady response of non-Newtonian blood flow through a stenosed artery in magnetic field. *J. Comput. Appl. Math.* **230** (2009), 243–259.
- [8] Jonášová, A.: *Computational modelling of hemodynamics for non-invasive assessment of arterial bypass graft patency*. Ph.D. thesis, University of West Bohemia, Pilsen, 2014.
- [9] Keslerová, R. and Kozel, K.: Numerical simulation of viscous and viscoelastic fluids flow by finite volume method. In: J. Fořt, J. Fürst, J. Halama, R. Herbin and F. Hubert (Eds.), *Finite volumes for complex applications VI — problems and perspectives*, pp. 589–596. Springer-Verlag Berlin Heidelberg, 2011.
- [10] Keslerová, R. and Kozel, K.: Numerical simulation of generalized Newtonian and Oldroyd-B fluids flow. In: J. Chleboun, K. Segeth, J. Šístek and T. Vejchodský (Eds.), *Proceedings of Programs and Algorithms of Numerical Mathematics 16*, pp. 112–117. IM ASCR, Prague, 2013.
- [11] LeVeque, R.: *Finite-volume methods for hyperbolic problems*. Cambridge University Press, 2004.
- [12] Matos, H. M. and Oliveira, P. J.: Steady flows of constant-viscosity viscoelastic fluids in a planar T-junction. *Journal of Non-Newtonian Fluid Mechanics* **213** (2014), 15–26.
- [13] Poole, R. J., Haward, S. J., and Alves, M. A.: Symmetry-breaking bifurcations in T-channel flows: effects of fluid viscoelasticity. *Procedia Engineering* **79** (2014), 28–34.
- [14] Rabby, M. G., Razzak, A., and Molla, M. M.: Pulsatile non-Newtonian blood flow through a model of arterial stenosis. *Procedia Engineering* **56** (2013), 225–231.
- [15] Vimmr, J., Jonášová, A., and Bublík, O.: Modelling of blood’s non-Newtonian behaviour in patient-specific aorto-coronary bypass grafts. In: *Proc. 10th World Congress on Computational Mechanics*, pp. 1634–1653. São Paulo, Blucher, 2014.