Radka Keslerová; Karel Kozel; Vladimír Prokop

Numerical solution of Newtonian flow in bypass and non-Newtonian flow in branching channels

In: Jan Chleboun and Karel Segeth and Tomáš Vejchodský (eds.): Programs and Algorithms of Numerical Mathematics, Proceedings of Seminar. Prague, May 28-31, 2006. Institute of Mathematics AS CR, Prague, 2006. pp. 137–142.

Persistent URL: http://dml.cz/dmlcz/702828

NUMERICAL SOLUTION OF NEWTONIAN FLOW IN BYPASS AND NON-NEWTONIAN FLOW IN BRANCHING CHANNELS*

R. Keslerová, K. Kozel, V. Prokop

Abstract

This paper deals with the numerical solution of Newtonian and non-Newtonian flows. The flows are supposed to be laminar, viscous, incompressible and steady. The model used for non-Newtonian fluids is a variant of the power-law. Governing equations in this model are incompressible Navier-Stokes equations. For numerical solution we could use artificial compressibility method with three stage Runge-Kutta method and finite volume method in cell centered formulation for discretization of space derivatives. The following cases of flows are solved: flow through a bypass connected to main channel in 2D and 3D and non-Newtonian flow through branching channels in 2D. Some 2D and 3D results that could have an application in the area of biomedicine are presented.

1. Mathematical model

The motivation for numerical solution of the fluid flow of Newtonian and non-Newtonian fluids arises in many applications, e.g. in the biomedicine, food industry, chemistry, glaciology etc. Many common fluids are non-Newtonian: paints, solutions of various polymers, food products. The main points of non-Newtonian behaviour are the ability of the fluid to shear thin or shear thicken in shear flows, the presence of non-zero normal stress differences in shear flows, the ability of the fluid to yield stress, the ability of the fluid to exhibit relaxation, the ability of the fluid to creep, see [1]. The solution of flows in branching channels and channels with bypass is important for modelling of blood flow in arteries. The study of blood flow in large and medium arteries is a very complex task because of the heterogeneous nature of the problem and the extreme complexity of blood and arterial wall dynamics. Although blood is actually a non-Newtonian suspension of cells in plasma, it is reasonable to model it as a Newtonian fluid in vessels greater than approximately 0.5 mm in diameter [2]. The occurring shear rates are in a range where non-Newtonian effects are only in minor significance to the flow parameters. This type of flow could be described by conservation laws of mass and momentum (Navier-Stokes equations), where the influence of exterior forces and heat exchange is not taken into account. In this case the model of a vessel is a tube with rigid walls. The pulsatile character of blood flow is not considered as well as the elasticity of arterial walls.

*This work was sponsored by Research Plan MSM 6840770010 and GA AS CR No. A 100190505.

First, we consider the Newtonian fluids. The system of 2D Navier-Stokes equations for Newtonian fluids in dimensionless conservative form has the form:

$$\tilde{R}W_t + F_x + G_y = \frac{\tilde{R}}{\mathrm{Re}}\Delta W, \qquad \tilde{R} = \mathrm{diag}\|0,1,1\|. \tag{1}$$

where the Reynolds number defined as $\mathrm{Re} = dw^*/\nu$ in 2D and $\mathrm{Re} = d_h w^*/\nu$ in 3D is an important parameter of the flow. Quantity w^* is a characteristic velocity (the speed of upstream flows), $\nu = \eta/\rho$ is the kinematic viscosity, d is a length scale (the width of the channel), $d_h = 4S/O$ is the hydraulic diameter, S is the area section of the duct and O is the wetted perimeter. In equation (1), $W = (p, u, v)^T$ is the vector of solution, $\tilde{R} = \mathrm{diag}\|0,1,1\|$, and $F = (u, u^2 + p, uv)^T, G = (v, uv, v^2 + p)^T$ denote inviscid fluxes, (u, v) is the dimensionless velocity vector ($u = u^*/q_\infty$, $v = v^*/q_\infty$), p denotes the dimensionless pressure ($p = p^*/\rho q_\infty^2$), t is the dimensionless time ($t = t^* q_\infty/l$), and q_∞ is defined as a velocity of incoming flow ($q_\infty = u^*$).

In the case of non-Newtonian fluids the power-law fluids are considered. The dominant difference from the Newtonian behaviour is shear thinning or shear thickening. From variety of power-law fluids we choose the simplest one:

$$\tau(\mathbf{e}) = 2\nu_0 |\mathbf{e}|^r \mathbf{e}, \tag{2}$$

where τ is the stress tensor, $\mathbf{e} = (e_{ij}),\ i, j = 1, 2$, is the strain tensor with components $e_{11} = u_x,\ e_{12} = e_{21} = (v_x + u_y)/2,\ e_{22} = v_y,\ |\mathbf{e}|$ denotes the Euclidean norm of the tensor, ν_0 is a positive constant related to the limit of generalized viscosity $\mu_g(\kappa)$ when $\kappa \to 0$, r is a constant of the model. The model captures the shear thinning fluid if $r \in (-1, 0)$, shear thickening fluid if $r > 0$, and $r = 0$ corresponds to the Newtonian fluid. For the non-Newtonian fluids the system of 2D Navier-Stokes equations and the continuity equation in two dimensional case written in the dimensionless conservative form reads

$$\tilde{R}W_t + F_x + G_y = \frac{\tilde{R}}{\mathrm{Re}}(R_x + S_y) \tag{3}$$

where $R = (0, g_{11}, g_{21})^T,\ S = (0, g_{12}, g_{22})^T,\ g_{ij} = 2|\mathbf{e}|^r e_{ij},\ i, j = 1, 2$, with components of e_{ij} defined above. The terms on the right-hand side can be expanded as follows

$$\begin{aligned} (g_{11})_x + (g_{12})_y &= 2|\mathbf{e}|^r_x u_x + |\mathbf{e}|^r_y (u_y + v_x) + |\mathbf{e}|^r \Delta u, \\ (g_{21})_x + (g_{22})_y &= |\mathbf{e}|^r_x (u_y + v_x) + 2|\mathbf{e}|^r_y v_y + |\mathbf{e}|^r \Delta v. \end{aligned} \tag{4}$$

Let us stress that subindices $_x$ and $_y$ denote partial derivatives with respect to x and y and that Δ stands for the 2D Laplacian. At the inlet the Dirichlet boundary condition for velocity vector $(u, v)^T$ is prescribed, at the outlet the pressure value is given. On the wall the zero Dirichlet boundary conditions for the components of velocity are used.

2. Numerical model

For further solution of the system of equations (1), the artificial compressibility method is used. The continuity equation is completed with the term p_t/a^2, where

$a^2 > 0$. The pressure satisfies the artificial equation of state: $p = \rho/\delta$, in which ρ is the artificial density, δ is the artificial compressibility, that is connected to the artificial speed of sound by relation $a = \delta^{-\frac{1}{2}}$, see [3]. Then system of governing equations has the form

$$W_t + F_x + G_y = \frac{\tilde{R}}{\mathrm{Re}} \left(R_x + S_y \right), \tag{5}$$

where $W = (p/a^2, u, v)^T$. System of equations (5) is solved by a three stage Runge-Kutta method with given steady boundary conditions. At the inlet an extrapolation of the pressure is used. At the outlet the value of the pressure is prescribed by $p = p_2$, where p_2 is the dimensionless value of the pressure, that is higher then the initial value of the pressure at the inlet to ensure pressure gradient. On the walls there are non-permeability and no-slip conditions. The multistage Runge-Kutta method is stabilized by the artificial viscosity term (Jameson's type, see [4]):

$$\begin{aligned} W_{i,j}^{n} &= W_{i,j}^{(0)} \\ W_{i,j}^{(r)} &= W_{i,j}^{(0)} - \alpha_r \Delta t \overline{R} W_{i,j}^{(r-1)}, \quad r = 1, \ldots, m, \\ W_{i,j}^{n+1} &= W_{i,j}^{(m)}, \quad m = 3, \end{aligned}$$

where W_{ij}^n denotes an approximation of W at grid point (x_i, x_j) and at a time $t = t_n$, $\Delta t = t_n - t_{n-1}$ is the time step, and

$$\overline{R} W_{i,j}^{(r-1)} = \tilde{R} W_{i,j}^{(r-1)} - D W_{i,j}^{n}.$$

The coefficients are $\alpha_1 = 0.5, \alpha_2 = 0.5, \alpha_3 = 1.0$ and the term DW_{ij}^n is described below. The numerical method is of the second order in time and space. The form of residual $\tilde{R} W_{i,j}^n$ depends on the method used for the space discretization, which is in this case the finite volume method in the cell centered formulation:

$$\tilde{R} W_{i,j} = \frac{1}{\mu_{ij}} \sum_{k=1}^{4} \left[\left(F_k^i - \frac{1}{Re} F_k^v \right) \Delta y_k - \left(G_k^i - \frac{1}{Re} G_k^v \right) \Delta x_k \right], \tag{6}$$

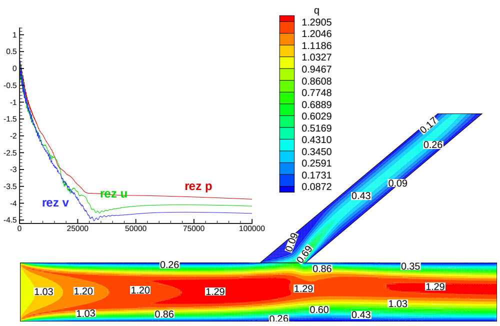
where $F^i = F, G^i = G$ are inviscid fluxes and $F^v = (0, u_x, v_x)^T, G^v = (0, u_y, v_y)^T$ are viscous fluxes, the index k corresponds to the side of a finite volume. The artificial viscosity term $DW_{i,j}^n$ depends in this case on the second derivatives of the pressure and is used to improve stability of the solution. The dissipative artificial viscosity term is constructed as follows:

$$
\begin{aligned}
DW &= D_x W + D_y W,\\
D_x W &= d_{i+\frac{1}{2},j} - d_{i-\frac{1}{2},j},\\
D_y W &= d_{i,j+\frac{1}{2}} - d_{i,j-\frac{1}{2}},\\
d_{i+\frac{1}{2},j} &= \frac{h_{i+\frac{1}{2},j}}{\Delta t}\epsilon^{(2)}_{i+\frac{1}{2},j}(W_{i+1,j} - W_{i,j}),\\
\nu_{i,j} &= \frac{|p_{i+1,j} - 2p_{i,j} + p_{i-1,j}|}{|p_{i+1,j}| + 2p_{i,j} + |p_{i-1,j}|},\\
\epsilon^{(2)}_{i+\frac{1}{2},j} &= \kappa^{(2)}\max(\nu_{i+1,j}, \nu_{i,j}),
\end{aligned}
$$

where $\kappa^{(2)}$ has to be chosen in order to achieve convergence of the method.

3. Numerical results

In this section we present steady numerical results obtained for the steady flow with the aid of the above methods. First, numerical results for channels with one entrance and two exit parts are presented. Figures 2 and 4 show the fluid velocity distribution for Reynolds number 1500 for non-Newtonian fluid. In Figures 3 and 5 the numerical results for the two-dimensional case (Newtonian fluids) and the convergence of the residuals of the vector $W = (p, u, v)^T$ are shown. The symbol q stands for the velocity magnitude, i.e. $q = \sqrt{u^2 + v^2}$. The other figures represent 3D flow for $\mathrm{Re} = 500$.



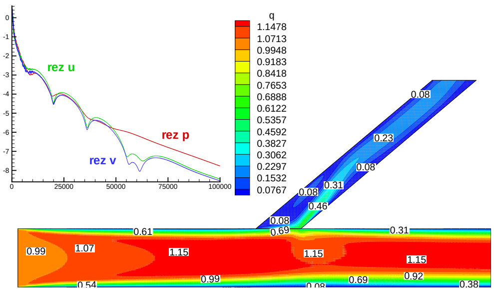


Fig. 1, 2: *Velocity magnitude distribution, non-Newtonian and Newtonian fluids,* $\mathrm{Re} = 1500$.

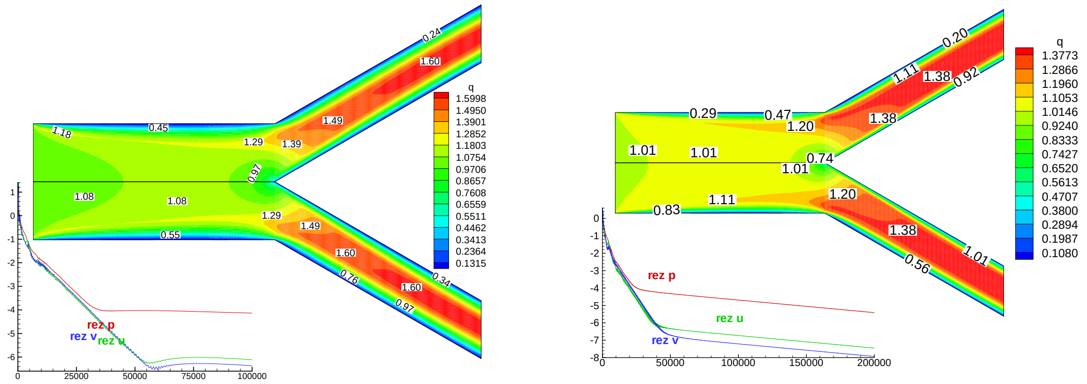


Fig. 3, 4: *Velocity magnitude distribution, non-Newtonian and Newtonian fluids,* Re = 1500.

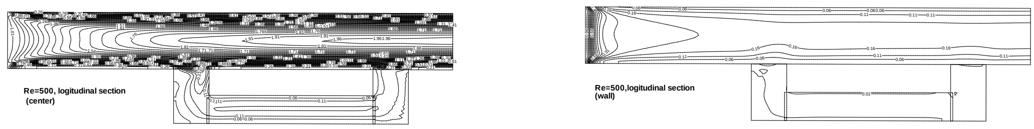


Fig. 5, 6: *Isolines of velocity in angular bypass for* Re = 500*, 3D case in the central plane xy, 3D case in the xy plane near the wall.*

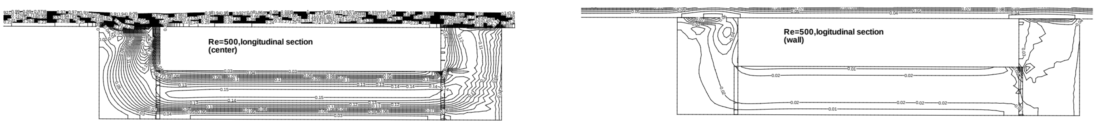


Fig. 7, 8: *Isolines of velocity in angular bypass for* Re = 500*, 3D case in the central plane xy, 3D case in the xy plane near the wall, details of regions.*

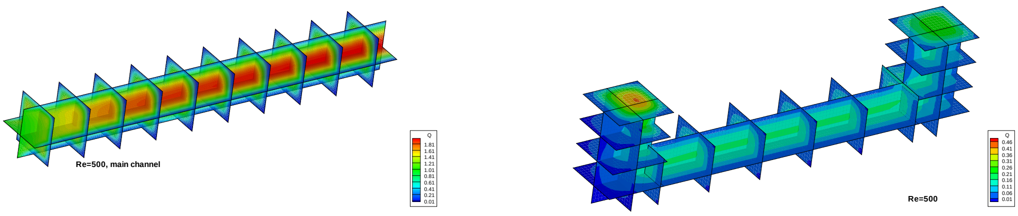


Fig. 9, 10: *The figure shows behaviour of flow in angular bypass for* Re = 500 *in the form of isolines of velocity, x-y-z cross section of the main channel and bypass.*

References

[1] K.R. Rajagopal: *Mechanics of non-Newtonian fluids.* In: G.P. Galdi, J. Nečas (eds.), Recent Developments in Theoretical Fluid Dynamics, Pitaman Research Notes in Math. **291**, Longman & Technical, Essex, 1993, 129–162.

[2] T. Taylor, T. Yamaguchi: *Three-dimensional simulation of blood flow in an abdominal aortic aneurysm, steady and unsteady flow cases.* J. Biomech. Engrg. **116**, 1994, 89–97.

[3] A.J. Chorin: *A numerical nethod for solving incompressible viscous flow problems.* Journal of Computational Physics **135**, 1997, 118–125.

[4] A. Jameson, W. Schmidt, E. Turkel: *Numerical solution of the Euler equations by finite volume methods using Runge-Kutta time-stepping schemes.* AIAA 14th Fluid and Plasma Dynamic Conference, June 23–25, 1981, Palo Alto, California, 1981, 1981–1259.

[5] R.J. LeVeque: *Numerical methods for conservation laws.* Birkhäuser Verlag, Basel, Switzerland, 1990.

[6] R. Dvořák, K. Kozel: *Mathematical modeling in aerodynamics.* CTU Prague, 1996.

[7] R. Keslerová, K. Kozel: *Numerical solution of 2D and 3D incompressible laminar flows through a branching channel.* In: Proc. Topical Problems of Fluid Mechanics, Prague, 2005, 55–58.