Vladimír Prokop; Karel Kozel

Numerical solution of steady and unsteady bypass flow

In: Jan Chleboun and Petr Přikryl and Karel Segeth (eds.): Programs and Algorithms of Numerical Mathematics, Proceedings of Seminar. Dolní Maxov, June 6-11, 2004. Institute of Mathematics AS CR, Prague, 2004. pp. 191–195.

Persistent URL: http://dml.cz/dmlcz/702795

NUMERICAL SOLUTION OF STEADY AND UNSTEADY BYPASS FLOW*

Vladimír Prokop, Karel Kozel†

Abstract

This paper deals with a problem of numerical solution of laminar viscous incompressible stationary and nonstationary flows through a vessel with bypass. One could describe these problems by using model of the Navier-Stokes equations and find a steady solution of an unsteady system by using a multistage Runge-Kutta method together with a time dependent artificial compressibility method. Nonstationary solution is achieved from initial stationary solution by prescribing of nonstationary outlet conditions. Some results of numerical solution of cardiovascular problems are presented: stationary and nonstationary 2D flows in a vessel and a bypass.

1. Mathematical model

In the cardiovascular system we could find many different types of vessels like large arteries, vessels of medium size and capillaries. They differ in diameter and in thickness and composition of the wall. In larger vessels the blood flow can be assumed to behave as an incompressible continuum. One can describe this type of flow using the system of momentum and continuity equation written in conservation form:

$$\rho\frac{\mathrm{D}\vec{w}}{\mathrm{D}t} - \nabla\cdot\tau = \rho\vec{f} \tag{1}$$

$$\nabla\cdot\vec{w} = 0 \tag{2}$$

where $\frac{\mathrm{D}\vec{w}}{\mathrm{D}t} = \frac{\partial\vec{w}}{\partial t} + w_i\frac{\partial\vec{w}}{\partial x_i}$, τ is the stress tensor of the fluid, $\vec{w}$ is the velocity vector and $\vec{f}$ is the vector of external forces, which is later not taken into account. The density of the fluid, ρ, is supposed to be constant in physiological conditions, although it depends on the red cells concentration. Important feature of the blood flow is pulsatility caused by the periodic motion of the heart. It is also known [4] that there is scarcely any turbulence in vessels except some special cases. The walls of a tube which is the model of a vessel are supposed to be rigid and the velocity vector $\vec{w}$ is null on them. The blood flow can be assumed to be laminar [4]. Indeed, in physiological conditions, the values of speed involved are low enough. Morover,

*This work was partly suported by grant GACR No. 201/02/0684 and Research Plan MSM 210000010.

†Czech Technical University, Faculty of Mechanical Engineering, Department of Technical Mathematics

generally, the periodicity of the flow, together with short length of vascular districts, do not give rise to fully developed turbulence. The Reynolds number $Re = \frac{dw^*}{\nu}$ is an important feature of the flow behaviour. The quantity w^* is characteristic velocity (the speed of upstream flows), $\nu = \mu/\rho$ is kinematic viscosity (μ is dynamic viscosity) and d is a lenght scale (the width of a channel). In large and medium human vessels, the Reynolds number ranges from 400 up to 10000. Elasticity of vessel tubes is not considered. The flow could be then described as viscous, incompressible, laminar and non-stationary in 2D by the system of the Navier-Stokes equations without influence of exterior forces and heat exchange. The system is written in conservative non-dimensional vector form rewritten from (1), (2)

$$\tilde{R}W_t + F_x + G_y = \tilde{R}\frac{1}{Re}\left(W_{xx} + W_{yy}\right), \tag{3}$$

where $W = (p, u, v)^T$ is the vector of solution, $\tilde{R} = \text{diag}\|0, 1, 1\|$ and $F = (u, u^2 + p, uv)^T, G = (v, uv, v^2 + p)^T$ denote inviscid fluxes, (u, v) is the velocity vector, p denotes the pressure. For upstream boundary conditions we use the velocity vector (u, v), along the walls the vector of velocity is equal to zero because of the viscosity of the fluid and impenetrability of the wall, downstream boundary condition is $p = p_2$, which should ensure the pressure gradient.

2. Numerical model

Solution of the system (3) is obtained using the method of artificial compressibility, then the equation of continuity is completed with the term $\frac{1}{a^2}p_t$, where $a^2 \in R^+$. The system (4) that is numerically solved, is the folloving

$$W_t + F_x + G_y = \tilde{R}\frac{1}{Re}\left(W_{xx} + W_{yy}\right), \tag{4}$$

where $W = (\frac{p}{a^2}, u, v)^T$. Finding solution one could solve unsteady system (4) by finite volume method and by time dependent method. System of equations (4) is solved by a three stage Runge-Kutta method and given boundary conditions. At the inlet an extrapolation of the pressure is used. At the outlet the value of the pressure is prescribed by the sinus function $p_2 = p_{20}(1 + \alpha \sin 2\pi\omega t)$, where ω is a frequency and α is an amplitude. The multistage Runge-Kutta method is stabilized by the artificial viscosity term (Jameson's type):

$$W^n_{i,j} = W^{(0)}_{i,j} \tag{5}$$

$$W^{(r)}_{i,j} = W^{(0)}_{i,j} - \alpha_r \Delta t \overline{R} W^{(r-1)}_{i,j}, (r = 1, \ldots, m) \tag{6}$$

$$W^{n+1}_{i,j} = W^{(m)}_{i,j}, m = 3, \tag{7}$$

where

$$\overline{R}W^{(r-1)}_{i,j} = RW^{(r-1)}_{i,j} - DW^{(r-1)}_{i,j} \tag{8}$$

and the coefficients $\alpha_1 = 0.5, \alpha_2 = 0.5, \alpha_3 = 1.0$, so the numerical method is of the second order in time and space. The form of the residual $RW^n_{i,j}$ depends on the method used for solving of the space derivatives:

$$RW_{i,j} = \frac{1}{\mu_{ij}} \sum_{k=1}^{4} \big[\big(F^i_k - \frac{1}{Re} F^v_k \big) \Delta y_k - \big(G^i_k - \frac{1}{Re} G^v_k \big) \Delta x_k \big], \tag{9}$$

where μ_{ij} is the volume of the finite volume, $F^i = F,\ G^i = G$ are the inviscid fluxes and $F^v = (0, u_x, v_x)^T,\ G^v = (0, u_y, v_y)^T$ are the viscous fluxes, the index k is corresponding to the side of a finite volume. The artificial viscosity term $DW^n_{i,j}$ depends in this case on the second derivatives of the pressure and is used for improving the stability of the solution.

3. Some numerical results

In this section we present steady numerical results achieved by using the numerical methods described above. The first figure shows results of a steady flow for $Re = 2000$. The other figures show an unsteady flow for $Re = 1000$. We start from the steady solution for a specific Reynolds number and using the same method as for the steady computation only changing the outlet pressure condition by the formula $p_2 = p_{20}(1 + \alpha \sin 2\pi\omega t)$, where the value of ω is equal 0.25, $p_{20} = 0.3$ and $\alpha = 0.3$, we recieve the results for an unsteady flow. In both, steady and unsteady, cases one could see zones of separation in the bypass after bifurcation and also in the domain after a contraction of a vessel. Results presented here are for 20 percent contraction of the vessel and for the bypasses which are about 40 or 30 percent of the diameter of the vessel. For an unsteady solution it is necessary to use $a \to \infty$ or computation in dual (artifitial) time.

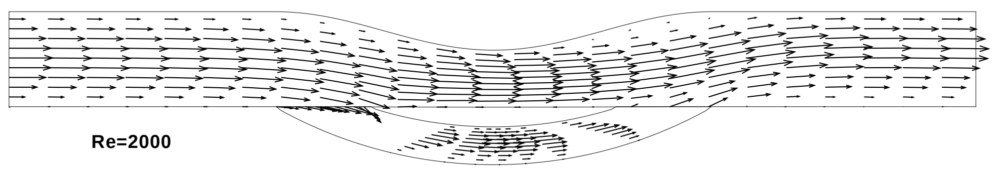


Fig. 1: *The figure shows the behaviour of the flow in the bypass for Re=2000: velocity vector field.*

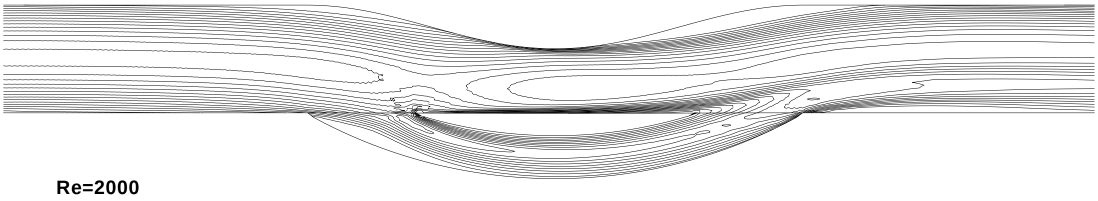
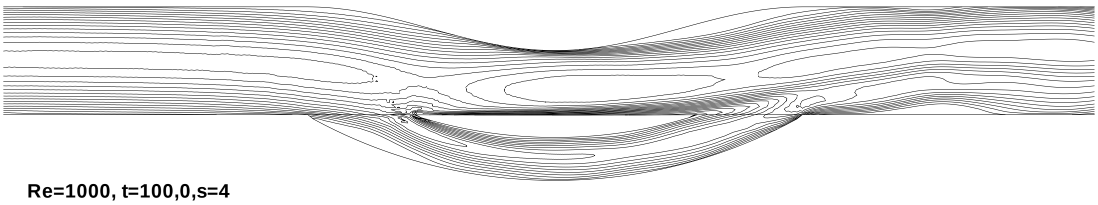
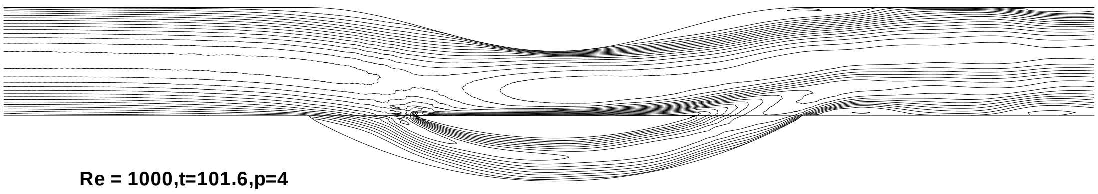
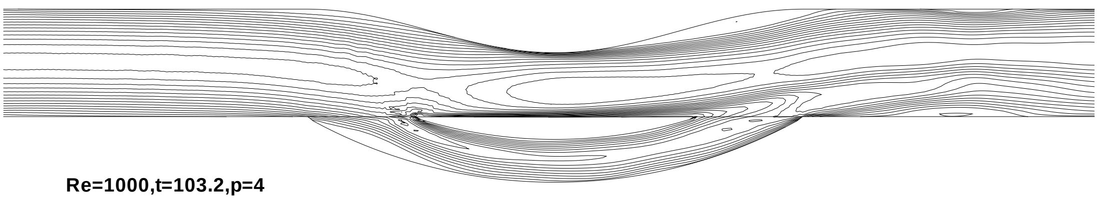


Fig. 2: *The figure shows the behaviour of the flow in the bypass for Re=2000: velocity isolines.*







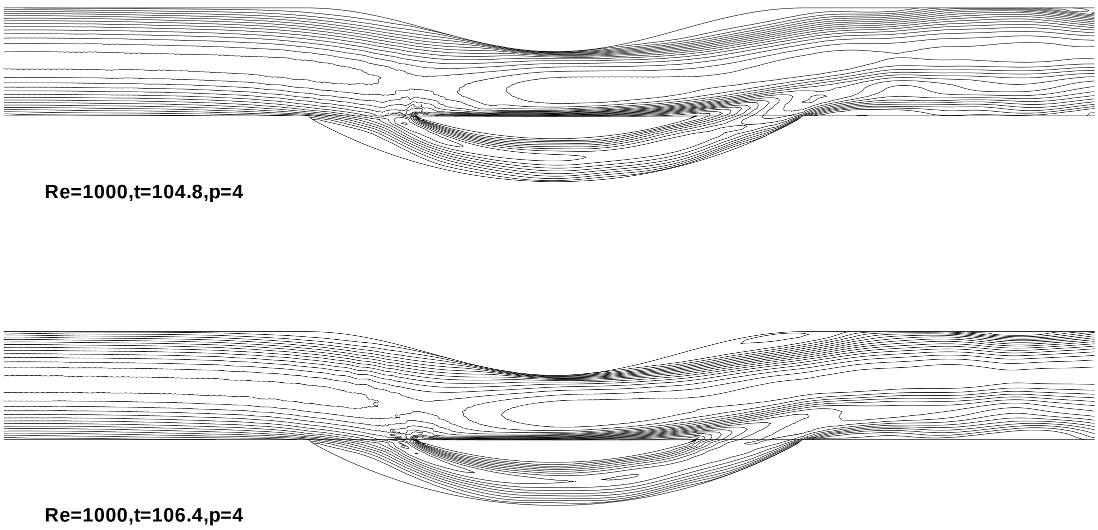


Fig. 3: *The above figures show the behaviour of the unsteady flow during two periods for Re=1000: velocity isolines.*

References

[1] S. Canic and D. Mirkovic. A hyperbolic system of conservation laws in modeling endovascular treatment of abdominal aortic aneurysm. *International Series of Numerical Mathematics*, 140, 2001.

[2] K. Kozel and V. Prokop. Numerical solution of incompresible viscous flows through a channel with cavitas and a channel with bypass. In *Proc. The Applied Mathematics and Informatics at Universities 2002*, number 2, pages 59–63, Trnava, September 2002.

[3] R. J. LeVeque. *Numerical Methods for Conservation Laws.* Birkhauser Verlag, 1990.

[4] A. Quarteroni, M. Tuveri, and A. Veneziani. Computational vascular fluid dynamics: Problems, models and methods. Unknown date and place of publishing.

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