PANM 12

Radka Keslerová; Karel Kozel

Numerical solution of 2D and 3D incompressible laminar flows through a branching channel

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. 94–101.

Persistent URL: `http://dml.cz/dmlcz/702781`

NUMERICAL SOLUTION OF 2D AND 3D INCOMPRESSIBLE LAMINAR FLOWS THROUGH A BRANCHING CHANNEL *

Radka Keslerová, Karel Kozel

Abstract

In this paper, we are concerned with the numerical solution of 2D/3D flows through a branching channel where viscous incompressible laminar fluid flow is considered. The mathematical model in this case can be described by the system of the incompressible Navier-Stokes equations and the continuity equation. In order to obtain the steady state solution the artificial compressibility method is applied. The finite volume method is used for spatial discretization. The arising system of ordinary differential equations (ODE) is solved by a multistage Runge-Kutta method. Numerical results for both 2D and 3D cases are presented.

1. Mathematical model

The motivation for numerical solution of the fluid flow in branching channels arises in many applications, e.g., in biomedicine, the solution of the blood flow in cardiovascular system is of interest. The study of the 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. Mathematical and numerical investigation of the blood circulatory system is one of the major challenges of the coming decades. During the 1970s, in vitro experiments were the main mode of cardiovascular investigations. Recently, the advances in computational fluid dynamics have lead to a significant breakthrough in vascular research. Physical quantities that are hard to measure in vivo can be computed for real geometries now (for details see, e.g., [4])

In this paper the system of Navier-Stokes equations for incompressible laminar flow is treated, which can be considered as a simplified mathematical model of the blood flow in a cardiovascular problem. The system of 2D Navier-Stokes equations and the continuity equation in two dimensional case written in the conservative form reads

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

where

$$W = (p,u,v)^T, \quad F = (u, u^2 + p, uv)^T, \quad G = (v, uv, v^2 + p)^T. \tag{2}$$

The following notation is used:

*This work was partly supported by grant No. 201/05/0005 and Research Plan MSM 68 40770003

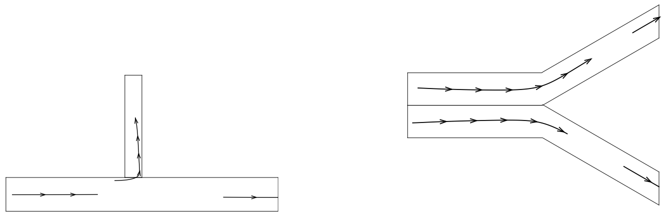

Fig. 1: *Considered forms of channel for 2D case.*

$(u, v)^T$	- velocity vector for 2D in dimensionless form
p	- kinetic pressure
Re	- Reynolds number defined as $\mathrm{Re} = \frac{q_\infty l}{\nu}$
ν	- the kinematic viscosity
l	- the height of the entrance
q_∞	- reference velocity value

At the inlet, the Dirichlet boundary condition for the 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 solution by the finite volume method

In what follows we are interested in steady state solutions. In such a case, the artificial compressibility method can be applied for the solution of the system (1), i.e.,

$$W_t + \tilde{F}_x + \tilde{G}_y = 0, \tag{3}$$

where

$$\tilde{F} = F - \frac{1}{\mathrm{Re}}\, F^v, \qquad \tilde{G} = G - \frac{1}{\mathrm{Re}}\, G^v,$$

and F, G are inviscid fluxes defined by (2), whereas F^v, G^v are viscous fluxes,

$$F^v = (0, u_x, v_x)^T, \quad G^v = (0, u_y, v_y)^T.$$

Eq. (3) is integrated over D_{ij} (D_{ij} is a finite volume cell, $\mu_{ij} = \iint\limits_{D_{ij}} \mathrm{d}x\mathrm{d}y$),

$$\iint\limits_{D_{ij}} W_t \mathrm{d}x\mathrm{d}y = -\iint\limits_{D_{ij}} \left(\tilde{F}_x + \tilde{G}_y\right) \mathrm{d}x\mathrm{d}y, \tag{4}$$

the mean value theorem is applied to the left-hand side of (4), and Green's theorem on the right-hand side of (4), so that

$$W_t \mid_{ij} = -\frac{1}{\mu_{ij}} \oint\limits_{\partial D_{ij}} \tilde{F}\mathrm{d}y - \tilde{G}\mathrm{d}x. \tag{5}$$

Next, we numerically approximate the integrals on the right hand side of (5) by

$$W_t \mid_{ij} = -\frac{1}{\mu_{ij}} \sum_{k=1}^{4} \tilde{F}_{ij,k}\Delta y_k - \tilde{G}_{ij,k}\Delta x_k, \tag{6}$$

where viscous fluxes in $\tilde{F}, \tilde{G}$ are conjucted using dual volumes.

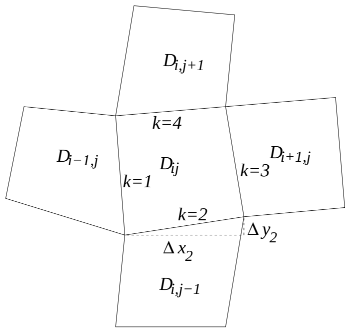

Fig. 2: *Finite volume cell.*

The ODE system (6) is time-discretized with the aid of a multistage Runge-Kutta method, i.e.,

$$\begin{array}{rcll} W_{ij}^{n} & = & W_{ij}^{(0)} & \\ W_{ij}^{(r)} & = & W_{ij}^{(0)} - \alpha_r \Delta t \overline{R} W_{ij}^{(r-1)} & \\ W_{ij}^{n+1} & = & W_{ij}^{(m)} & \qquad r = 1, \ldots, m, \end{array} \tag{7}$$

where $m = 3$, $\alpha_1 = \alpha_2 = 0.5$, $\alpha_3 = 1$, $\overline{R}W_{ij}^{n} = RW_{ij}^{n} - DW_{ij}^{n}$, and the steady residual RW_{ij} is defined by

$$RW_{ij} = \frac{1}{\mu_{ij}} \sum_{k=1}^{4} \left[\left(F_k^i - \frac{1}{\mathrm{Re}} F_k^v \right) \Delta y_k - \left(G_k^i - \frac{1}{\mathrm{Re}} G_k^v \right) \Delta x_k \right]. \tag{8}$$

The added artificial viscosity term DW_{ij} of Jameson's type (for details see, e.g., [3]) is defined as

$$DW_{ij} = E\left[\gamma_i \left(W_{i+1,j} - 2W_{ij} + W_{i-1,j}\right) + \gamma_j \left(W_{i,j+1} - 2W_{ij} + W_{i,j-1}\right)\right] \tag{9}$$

$$E = \mathrm{diag}\|0, \epsilon_1, \epsilon_2\|, \quad \epsilon_1, \epsilon_2 \in \Re, \quad \gamma_i = \max(\gamma_{i1}, \gamma_{i2}), \quad \gamma_j = \max(\gamma_{j1}, \gamma_{j2}),$$

$$\gamma_{i1} = \frac{| p_{i+1,j} - 2p_{ij} + p_{i-1,j} |}{| p_{i+1,j} + 2p_{ij} + p_{i-1,j} |}, \qquad \gamma_{i2} = \frac{| p_{ij} - 2p_{i-1,j} + p_{i-2,j} |}{| p_{ij} + 2p_{i-1,j} + p_{i-2,j} |},$$

$$\gamma_{j1} = \frac{| p_{i,j+1} - 2p_{ij} + p_{i,j-1} |}{| p_{i,j+1} + 2p_{ij} + p_{i,j-1} |}, \qquad \gamma_{j2} = \frac{| p_{ij} - 2p_{i,j-1} + p_{i,j-2} |}{| p_{ij} + 2p_{i,j-1} + p_{i,j-2} |}.$$

In order to satisfy the stability condition, the time step is chosen as (for details see, e.g., [1]):

$$\Delta t = \min_{i,j,k} \frac{\text{CFL } \mu_{ij}}{\rho_A \Delta y_k + \rho_B \Delta x_k + \frac{2}{\text{Re}} \left(\frac{(\Delta x_k)^2 + (\Delta y_k)^2}{\mu_{ij}} \right)}, \tag{10}$$

$$\rho_A =| \hat{u} | + \sqrt{\hat{u}^2 + 1} \qquad \rho_B =| \hat{v} | + \sqrt{\hat{v}^2 + 1},$$

and $| \hat{u} |, | \hat{v} |$ are the maximal values of the components of velocity inside the computational domain; the definition of $\Delta x_k, \Delta y_k$ is shown in Fig. 2.

The computation is performed until the value of the L^2-norm of residual satisfies Rez $W_{ij}^n \leq \epsilon_{ERR}$ with ϵ_{ERR} small enough (MN denotes the number of grid cells in the computational domain), where

$$\text{Rez } W_{ij}^n = \sqrt{\frac{1}{MN} \sum_{ij} \left(\frac{W_{ij}^{n+1} - W_{ij}^n}{\Delta t} \right)^2}. \tag{11}$$

3. Numerical results

In this paper we present the numerical results for channels with one entrance and two exit parts. The computation of both two-dimensional and three-dimensional cases was performed. First, in Figs. 3 and 4 the numerical results for the two-dimensional case are shown. Fig. 3 shows the fluid velocity distribution for Reynold's number 1000 in the channel of the form of reverse T and the convergence of the residuals of the vector $W = (p, u, v)^T$. Fig. 4 shows the velocity isolines for Reynold's number 1500 for the symmetric branching channel of the form Y. By the symbol q, the velocity magnitude is denoted, i.e. $q = \sqrt{u^2 + v^2}$.

Next, the computation for fully three-dimensional fluid flow in two distinct cases was performed. Figs. 5, 6 show the velocity isolines in the cross-sections of the branching channels for Reynold's number 300. Figs. 7, 8 show the velocity isolines in the cross-sections of the symmetric branching channel for Reynold's number 300. In Fig. 7, the convergence of the residuals of the components of the vector $W = (p, u, v, w)^T$ is shown. Symbol q denotes the velocity magnitude for the three-dimensional case, i.e., $q = \sqrt{u^2 + v^2 + w^2}$.

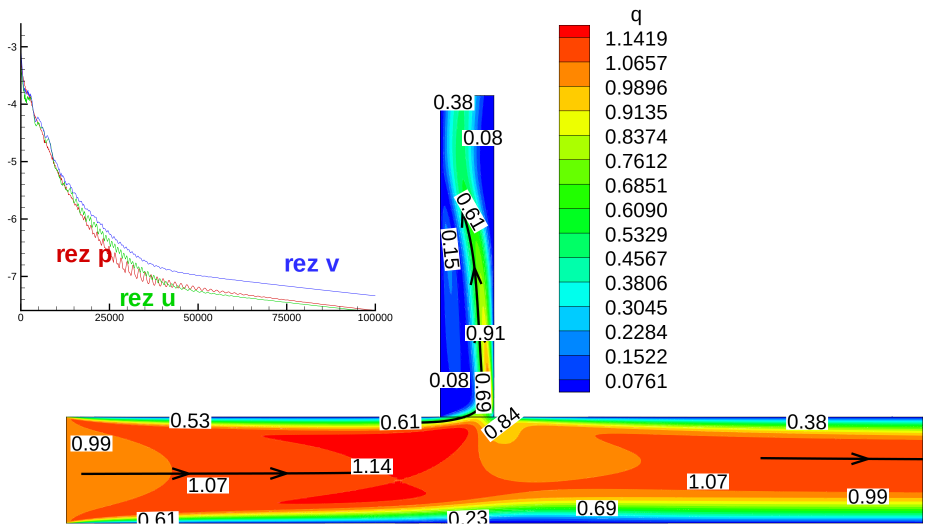

Fig. 3: *Velocity magnitude distribution in two dimensional channel of reverse T form (Reynolds number Re = 1000).*

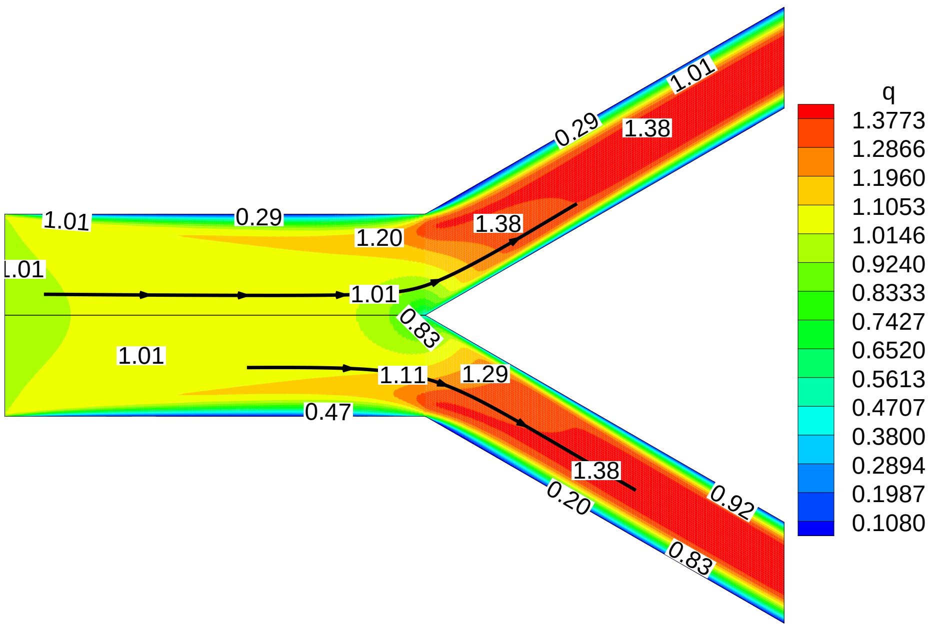

Fig. 4: *Velocity magnitude distribution in two dimensional symmetric channel of Y form (Reynolds number Re = 1500).*

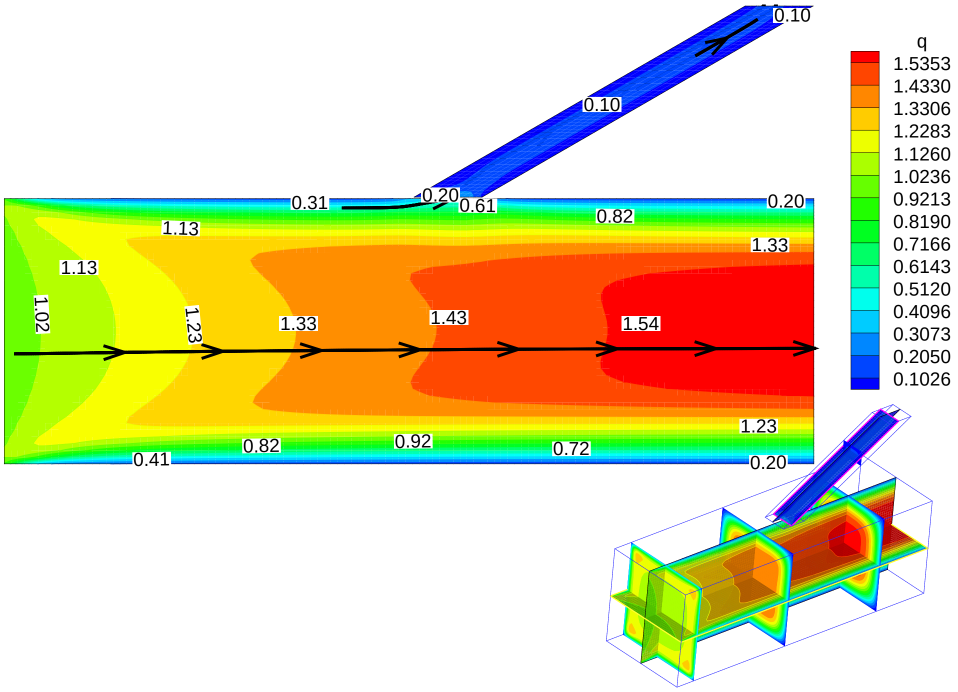

Fig. 5: *Velocity magnitude distribution in the central cut of the three dimensional channel (Reynolds number Re = 300).*

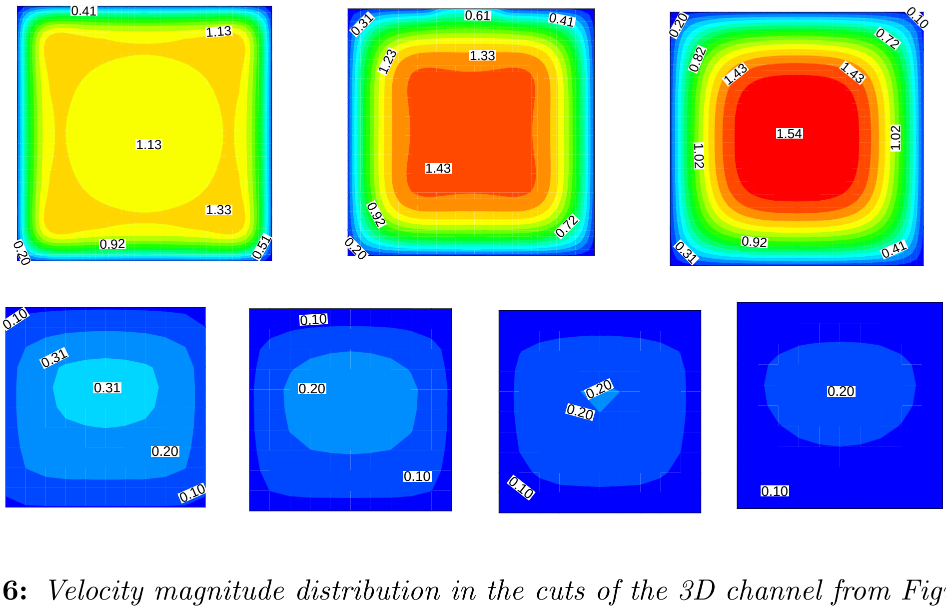

Fig. 6: *Velocity magnitude distribution in the cuts of the 3D channel from Figure 5.*

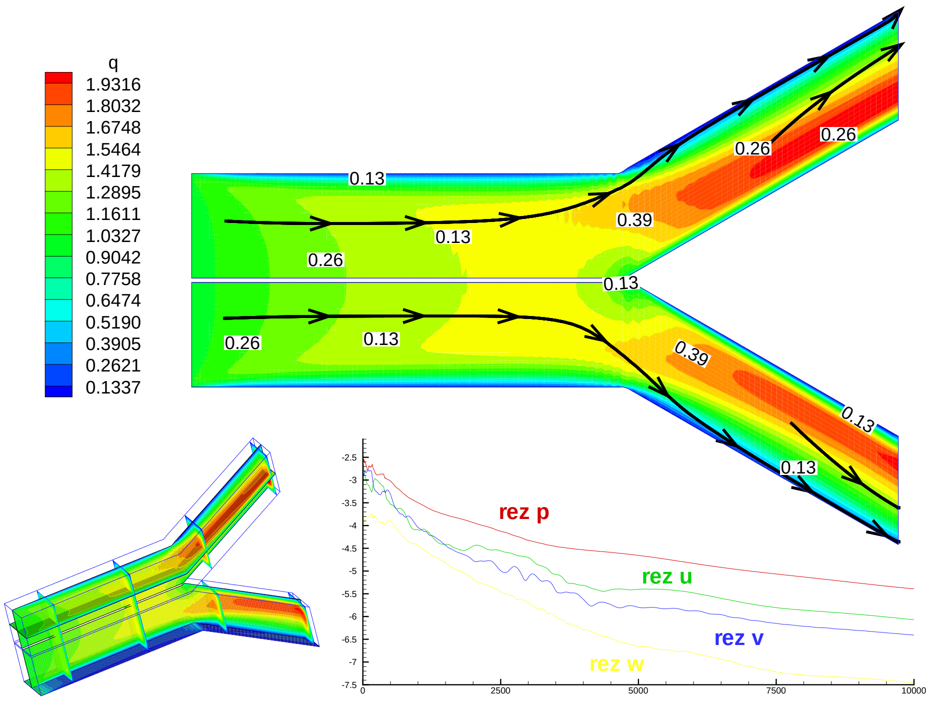

Fig. 7: *Velocity magnitude distribution in the central cut of the three dimensional channel (Reynolds number Re = 300).*

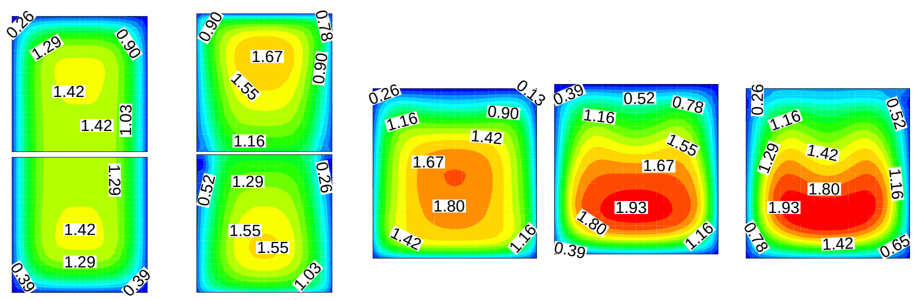

Fig. 8: *Velocity magnitude distribution in the cuts of the 3D channel from Figure 7.*

4. Conclusion

A numerical model for the simulation of fluid flow in a branching channel with one entrance and two exit parts for two-dimensional and three-dimensional cases was developed. However, the method was applied for several different types of channel configurations. The presented results can be useful as a blood flow approximation but several significant simplifications made in the model should be mentioned. First, the general blood flow can be characterized rather by the non-Newtonian fluid flow model (see [4]) so that the use of the Newtonian fluid flow model is questionable. Second, the vessel's walls cannot be considered as fixed walls and thus the computational domain deformations needs to be taken into account. This phenomenon can be treated, e.g., by the arbitrary Lagrangian-Eulerian (ALE) method (for ALE description see, e.g., [7], for practical applications see, e.g., [5], [6]). Moreover, the wall dynamics needs to be modelled and coupled with the fluid flow model. However, the presented results can be still used as an approximation of the problem, which can provide useful information, e.g., the fluid flow character, the appearance of separation zones, pressure distribution, etc. The applied method serves as a starting point and will be further developed. From this point of view it is necessary to test the method on some standard problems and compare the results with the other available data (see e.g. [8]). Moreover, the presented method can be extended and applied to more realistic blood flow model, which is the subject of our future work.

References

[1] R. Dvořák, K. Kozel. *Mathematical Modelling in Aerodynamics.* CTU Prague 1996. (In Czech.)

[2] K. Kozel, R. Keslerová, V. Prokop. Numerical Solution of Icompressible Laminar Flows through a Branching Channel. In Proceedings of *International Conference of Applied Mathematics and Informatics at Universities 2002*, Trnava, Slovak Republic

[3] R. C. Swanson, E. Turkel. Multistage Schemes With Multigrid for Euler and Navier-Stokes Equations. *NASA Technical Paper 3631*, 1997.

[4] A. Quarteroni, M. Tuveri, A. Veneziani. Computational vascular fluid dynamics: Problems, models and methods. *Computing and Visualization in Science 2*, 2000, pp. 163–197.

[5] P. Sváček, M. Feistauer, J. Horáček. Numerical Simulation of a Flow Induced Airfoil Vibrations. In *Flow Induced Vibrations*, volume 2, pp. 57–62. Ecole Polytechnique, 2004.

[6] P. Sváček, M. Feistauer, J. Horáček. Numerical simulation of a flow induced airfoil vibrations with large amplitudes. Preprint No. MATH-knm-2005/1, pp. 29, Faculty of Mathematics and Physics, Charles University Prague 2005.

[7] T. Nomura, T. J. R. Hughes. An arbitrary Lagrangian-Eulerian finite element method for interaction of fluid and a rigid body. *Computer Methods in Applied Mechanics and Engineering 95*, 1992, pp. 115–138

[8] K. Kozel, V. Prokop. Numerical Simulation of Some Cardiovascular Problems. In Proceedings of *Topical Problems of Fluid Mechanics, 2003*, pp. 57-60.