DOI: <https://doi.org/10.14311/TPFM.2022.018>

NUMERICAL VALIDATION OF A SIMPLE IMMERSED BOUNDARY SOLVER FOR BRANCHED CHANNELS SIMULATIONS

A. Lancmanová¹, T. Bodnár^{1,2}, R. Keslerová¹

¹ Faculty of Mechanical Engineering, Czech Technical University in Prague,
Karlovo náměstí 13, 121 35 Prague 2, Czech Republic.

² Institute of Mathematics, Czech Academy of Sciences,
Žitná 25, 115 67 Prague 1, Czech Republic.

Abstract

This contribution reports on an ongoing study of incompressible viscous fluid flow in two dimensional branched channels. A new finite difference solver was developed using a simple implementation of an immersed boundary method to represent the channel geometry. Numerical solutions obtained using this new solver are compared with outputs of an older finite volume code working on classical wall fitted structured multiblock grid. Besides of the comparative evaluation of obtained solution, the aim is to verify whether the immersed boundary method is suitable (accurate and efficient enough) for simulations of flow in channels with complicated geometry where the the grid generation might be challenging.

Keywords: branching channel, incompressible Navier-Stokes, finite difference, MacCormack, artificial compressibility

1 Introduction

Fluid flow in branching (sometimes also called bifurcating or dividing) channels is of interest in many technical [3] or biomedical applications [15], [16], [9]. The air flow in lungs or blood flow in circulatory system of living organisms can be taken as an example and motivation for our study [8], [17], [13]. One of the challenges in simulating this kind of flows where the domain occupied by the fluid has complicated geometry with several levels of (almost fractal like) branching. Although it is possible to discretize such kind of geometry by standard grids (structured or unstructured), the grid generation might be difficult to guarantee accurate and consistent numerical results of fluid flow simulations.

Our final aim is to develop a mathematical models, numerical methods and solvers for biomedical applications. For this purpose various non-standard artificial boundary conditions need to be implemented and tested for number of complicated channel configurations. Thus we have decided to build a specialized simple test code on which such boundary conditions, numerical methods and mathematical models can easily be tested, even for non-trivial branched channels. To achieve this objective, we have chosen a finite difference method to be applied on regular structured grid, where the specific channel geometry will only be represented using the immersed boundary method.

The main aim in this present work is to compare the results of a standard finite volume based method [10], [9] which used the wall fitted grid, with a much simpler finite difference code working on regular Cartesian grid [12] while employing a generic implementation of immersed boundary method. As a test case a channel with single branch inclined at different angles was chosen, similar to geometry used in [16] or [14]. The results of both codes are compared to see whether the simple finite difference and immersed boundary method based code can match the essential flow characteristics resolved by the older standard finite volume code used in some of our previous studies [9], [2].

2 Mathematical Model

The mathematical model and the choice of the computational test case is motivated by the previously mentioned biomedical applications. Although both numerical methods can easily be implemented in fully three-dimensional setting, most of the description below is written for two-dimensional case because only such cases were solved numerically. Unless noted otherwise, the

below provided description is common for both, the finite-difference and finite-volume methods used in our numerical simulations.

2.1 Governing equations

The incompressible Navier-Stokes equations can be written in a conservative vector form:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) = -\frac{1}{\rho} \nabla p + \nu \Delta \mathbf{u}, \quad (1)$$

where ρ is the (constant) density, ν is the (constant) kinematic viscosity and p , $\mathbf{u}$ is the pressure and velocity field respectively. The velocity field obeys the incompressibility (divergence-free) constraint ($\nabla \cdot \mathbf{u} = 0$).

2.2 Computational domain

For the immersed boundary implementation of finite-difference method, the two-dimensional (2D) computational domain was chosen as a rectangle in $x-z$ plane with dimensions $30D \times 10D$. The numerical simulations were performed on a structured, orthogonal (Cartesian) grid with different number of equidistant nodes. The standard grid had 1200×200 cells.

The coordinate system has been chosen to have the origin at the edge of branching ($x-z$ coordinates). The whole domain as well as the channel geometry is shown in Fig.1. The width (diameter) of the main horizontal channel is denoted by D and the width of the inclined (oblique) branch inclined at the angle α was chosen to be $D/2$. The same configuration was kept for all

Figure 1: Computational domain of a branched pipe.

simulations, just changing the angle α by setting it to values 30° , 60° , 90° , 120° and 150° .

For finite volume simulations, just the interior of the channel (marked by blue color in Fig.1) was used to construct the grid. See section 4 for details.

2.3 Boundary conditions

The boundary setup was chosen as simple as possible in order to allow for simple extension for non-Newtonian and turbulent flows, where the velocity profiles are not a-priori known even for the simple channel with Poiseuille like flow. Therefore we have opted for the flow to be defined by pressure drop to be prescribed between the inlet and outlet parts of the boundary. So only different values of pressure were prescribed at different inlet/outlet parts of the boundary. Otherwise the homogeneous Neumann condition was prescribed for velocity components on those part of boundary to mimic a fully developed flow. On the channel wall of course the no-slip, i.e. homogeneous Dirichlet condition $\mathbf{u} = (0, 0)$ was prescribed for velocity.

To establish a pressure drop between inlet and outlet of the channel, the pressure was set to zero at outlet, and a suitable (positive) value of pressure was set at the inlet. In order to achieve some realistic flow conditions, we have chosen all conditions similar to flow of blood in common carotid artery. From its diameter $D = 6mm$, fluid viscosity and channel length (without considering the branch) it was possible to find suitable inlet pressure from Poiseuille solution, to achieve similar velocity as it's found in the real blood vessel of the same size. This led us to the choice of inlet pressure $p_{in} = 60Pa$ while at the outlets we prescribed pressures $p_A = p_B = 0$.

For the immersed boundary method the velocity is set to zero inside the part of the domain occupied by the solid material, so no special treatment is needed on the channel walls for pressure or velocity. See section 3.2 for details of the implementation.

3 Numerical Methods

As it was mentioned in the Introduction, two different numerical methods were used in this study. They will be described separately in the Section 3.2 and 3.3 respectively. Both methods share the *artificial compressibility* approach used to resolve pressure and to enforce the divergence-free constraint.

3.1 Artificial compressibility method

Foundations of this method can be tracked down to 1967 when A. J. Chorin published the method in [4] and [5]. The principle of artificial compressibility method consists in adding a time derivative of pressure to the continuity equation leading to

$$\frac{1}{\beta^2} \frac{\partial p}{\partial t} + \nabla \cdot \mathbf{u} = 0 \quad (2)$$

where $\beta^2 \in \mathbb{R}^+$ represents the coefficient of artificial compressibility. The pressure then can be evaluated from this equation in the same manner as the velocity can be calculated from the momentum equations (1). More details on this method can be found for example in classical book [7], paper [18], or in some of our previous works [10], [9].

In order to describe (and implement) the numerical methods in a simple way, it's convenient to rewrite the governing system, including the artificial compressibility method, in vector form. Starting from the complete modified governing system (1)-(2) written in conservative form (the same in 2D and 3D). The two-dimensional (in $x - z$ coordinates) version of the system can be written in vector form

$$D\mathbf{W}_t + \mathbf{F}_x + \mathbf{H}_z = \mathbf{R}_x + \mathbf{T}_z, \quad (3)$$

where $D = \text{diag}(\beta^{-2}, 1, 1)$ represents diagonal matrix including the artificial compressibility parameter β , $\mathbf{W} = \text{col}(p, u, w)$ is the vector of unknowns, $\mathbf{F}, \mathbf{H}$ and $\mathbf{R}, \mathbf{T}$ are inviscid and viscous fluxes respectively defined by (4) and (5).

$$\mathbf{F} = \begin{pmatrix} u \\ u^2 + p/\rho \\ u w \end{pmatrix}, \quad \mathbf{H} = \begin{pmatrix} w \\ w u \\ w^2 + p/\rho \end{pmatrix} \quad (4)$$

$$\mathbf{R} = \begin{pmatrix} 0 \\ \nu u_x \\ \nu w_x \end{pmatrix}, \quad \mathbf{T} = \begin{pmatrix} 0 \\ \nu u_z \\ \nu w_z \end{pmatrix} \quad (5)$$

3.2 Finite-difference method

The newly developed finite-difference code relies on immersed boundary method to represent the computational domain and on the MacCormack scheme to discretize the governing equations.

Immersed boundary method When applying an immersed boundary method, we select usually a rectangular domain that contains both the fluid region and also the bounding solid. The domain contains regular structured grid and some grid points are in the fluid region and some are in the solid region. For the calculation, we introduce a solid function d that takes the values according to where it is located in the area

- $d = 1$ if point is located in fluid region,
- $d = 0$ if point is located in solid region.

Figure 2: Immersed boundary implementation.

MacCormack scheme It is a specific version of the Lax-Wendroff scheme, reformulated in predictor-corrector form [7]. The MacCormack scheme uses one-sided differences for discretization of spatial derivatives of inviscid fluxes. Different backward/forward approximation is used in predictor/corrector, which finally combined gives a central approximation of the discretized derivatives. The viscous fluxes are always discretized in the central manner in both predictor and corrector stages.

To describe the MacCormack scheme, it's better to start from a rearranged equation (3), where all terms except the time derivative are placed on the right hand side.

$$\mathbf{W}_t = \mathbf{D}^{-1} [-(\mathbf{F}_x + \mathbf{H}_z) + \mathbf{R}_x + \mathbf{T}_z]. \quad (6)$$

Now in order to update in time the values of the vector $\mathbf{W}^n$ to $\mathbf{W}^{n+1}$ the aim is to construct an approximation of $\mathbf{W}_t$ from (6). This approximation is built differently in predictor (e.g. using backward differences) and in corrector (using forward differences). Then the final update is performed using linear combination of the two values obtained. This procedure is formalized in the following steps:

Predictor step — The spatial derivatives of inviscid fluxes $\mathbf{F}$, $\mathbf{H}$ in (6) are discretized using backward differences, while central differencing is used for viscous fluxes $\mathbf{R}$ and $\mathbf{T}$.

$$\left(\frac{\partial \mathbf{W}}{\partial t} \right)_{i,j}^n = \mathbf{D}^{-1} \left[- \left(\frac{\mathbf{F}_{i,j}^n - \mathbf{F}_{i-1,j}^n}{\Delta x} + \frac{\mathbf{H}_{i,j}^n - \mathbf{H}_{i,j-1}^n}{\Delta z} \right) + \left(\frac{\mathbf{R}_{i+1,j}^n - 2\mathbf{R}_{i,j}^n + \mathbf{R}_{i-1,j}^n}{\Delta x^2} + \frac{\mathbf{T}_{i,j+1}^n - 2\mathbf{T}_{i,j}^n + \mathbf{T}_{i,j-1}^n}{\Delta z^2} \right) \right], \quad (7)$$

where values of all variables on the right side are known at the current time level n . Using the approximate value $\left(\frac{\partial \mathbf{W}}{\partial t} \right)_{i,j}^n$, the preliminary (marked by tilde) values of unknown $\mathbf{W}^{n+1}$ are predicted from the first two terms of the corresponding Taylor series, i.e. using explicit Euler time-stepping:

$$\widetilde{\mathbf{W}}_{i,j}^{n+1} = \mathbf{W}_{i,j}^n + \Delta t \left(\frac{\partial \mathbf{W}}{\partial t} \right)_{i,j}^n, \quad (8)$$

where the value of the first term on the right hand side is known at current time level and the second term was previously evaluated from (7). The obtained auxiliary values of $\widetilde{\mathbf{W}}^{n+1}$ are used in the corrector step.

Corrector step — Here the predicted values $\widetilde{\mathbf{W}}^{n+1}$ are used to calculate another approximation of $\mathbf{W}_t$ from (6), but this time applying the forward in space differentiation of inviscid fluxes $\mathbf{F}$ and $\mathbf{H}$ while central differencing is used again for viscous fluxes $\mathbf{R}$ and $\mathbf{T}$.

$$\left(\frac{\partial \widetilde{\mathbf{W}}}{\partial t} \right)_{i,j}^{n+1} = \mathbf{D}^{-1} \left[- \left(\frac{\widetilde{\mathbf{F}}_{i+1,j}^{n+1} - \widetilde{\mathbf{F}}_{i,j}^{n+1}}{\Delta x} + \frac{\widetilde{\mathbf{H}}_{i,j+1}^{n+1} - \widetilde{\mathbf{H}}_{i,j}^{n+1}}{\Delta z} \right) + \left(\frac{\widetilde{\mathbf{R}}_{i+1,j}^{n+1} - 2\widetilde{\mathbf{R}}_{i,j}^{n+1} + \widetilde{\mathbf{R}}_{i-1,j}^{n+1}}{\Delta x^2} + \frac{\widetilde{\mathbf{T}}_{i,j+1}^{n+1} - 2\widetilde{\mathbf{T}}_{i,j}^{n+1} + \widetilde{\mathbf{T}}_{i,j-1}^{n+1}}{\Delta z^2} \right) \right] \quad (9)$$

Again all the terms on the right hand side can easily be evaluated from the known values $\widetilde{\mathbf{W}}_{i,j}^{n+1}$.

Variables update — Having now the two approximate values of $\mathbf{W}_t$ from predictor and corrector steps, the final approximation can be built as an average of the two values, representing an approximation of $\mathbf{W}_t$ at the time level $n + 1/2$.

$$\left(\frac{\partial \mathbf{W}}{\partial t}\right)_{i,j}^{n+1/2} = \frac{1}{2} \left[\left(\frac{\partial \mathbf{W}}{\partial t}\right)_{i,j}^n + \left(\frac{\partial \widetilde{\mathbf{W}}}{\partial t}\right)_{i,j}^{n+1} \right] \quad (10)$$

Finally the values of the unknown variables vector $\mathbf{W}^{n+1}$ can be obtained by making a forward time-step of the length Δt from $\mathbf{W}^n$ using the approximate time-derivative $\mathbf{W}_t^{n+1/2}$.

$$\mathbf{W}_{i,j}^{n+1} = \mathbf{W}_{i,j}^n + \Delta t \left(\frac{\partial \mathbf{W}}{\partial t}\right)_{i,j}^{n+1/2} \quad (11)$$

In principle, the MacCormack method is thus similar to Heun's method for solution of ordinary differential equations. Also here it leads to increased (second order) accuracy with respect to time. The spatial accuracy of MacCormack method is the same (second order) as for the Lax-Wendroff scheme due to use of central approximation of all spatial derivatives [6].

It's good to note, that the choice of backward differencing in predictor and forward differencing in corrector steps was arbitrary and it could have been chosen the other way round and it shouldn't affect the results [7], [1].

3.3 Finite-volume method

The finite-volume method used in this study was described in detail in several of our previous publications, like for example [9], [10] or [2], so only basic information is given here. The implementation adopted for our code uses semi discrete approach with finite-volume discretization of spatial derivatives on structured grid with quadrilateral cells. The inviscid fluxes are discretized in central way on primary cells, while the viscous fluxes are approximated on dual, diamond shaped, cells. After discretization in space, a system of ordinary differential equations is obtained for evolution in time of the cell averaged quantities. The integration in time is performed using an explicit Runge-Kutta multistage scheme.

4 Numerical Simulations and Results

The numerical results shown in this section represent just a small portion of the large number of simulations performed. The main aim of the selection shown here is to demonstrate the main conceptual differences between the classical approximation and the immersed boundary representation of the computational geometry. The second goal is to show how the finite-difference immersed boundary method captures the main flow features as well as the smaller scale details.

The details of the finite-difference (FDM) cartesian grid and finite-volume (FVM) wall fitted grid is shown in Fig. 3.

Figure 3: Grid structure comparison - FDM grid (in blue) , FVM grid (in purple)

One of the main problems in constructing structured wall fitted grids for branching channels can be sudden changes and high distortion of grid cells. This problem is absent in Cartesian regularly

spaced grids, where even very simple numerical methods can achieve their highest accuracy. On the other hand, an accurate representation of domain geometry in immersed boundary method can be questionable as it can be seen in Fig. 4.

Figure 4: Velocity magnitude computed using FDM and immersed boundary method.

Evidently the immersed boundary approximation of oblique parts of the channel might cause some problems, while the parts of boundaries parallel to grid lines are well represented. This may result in different accuracy for different channels (branching angles). The same part shown with FVM grid and solution in Fig. 5.

Figure 5: Velocity magnitude computed using FVM on wall fitted grid.

Pressure values and character of the pressure field is dictated by the prescribed boundary conditions. In our setting the pressure is set to zero at outlet (right boundary) of the main channel as well as at the outlet of the branch (upper boundary), while the value $p = 60Pa$ is prescribed at the inlet (left boundary). This forms a pressure drop between inlet and outlet parts of the channel, leading to the development of the flow. The pressure field has almost linear decay along the main channel as well as along the thinner branch with some more complicated patterns in the vicinity of edges (corners) in the branching region, which is shown in Fig 6.

Finer scale flow features, including vortices, are resolved in the branching region, which can be seen in the Fig. 6, where the streamlines are drawn in the branching region.

Figure 6: Pressure field and streamlines in detail (FDM).

Some interesting features are revealed also by the contours of velocity magnitude shown in Fig. 7. The velocity in the thinner branch of the channel is higher for the case $\alpha = 90^\circ$, leading to higher flow rate in that branch and consequently also to higher velocities in the main channel. Further results for other geometries (angles α) as well as the results obtained using the FVM are showing similar features.

Figure 7: Velocity magnitude for $\alpha = 30^\circ$ and $\alpha = 90^\circ$ (FDM).

5 Conclusions & Remarks

This contribution shows some of the first comparative tests performed to validate our new finite-difference immersed boundary code. The following points briefly summarize the main findings of this ongoing work:

- The formally second order MacCormack scheme was successfully implemented to replace the previously used first order Lax-Friedrichs method reported in [11], [12]. This should be important in our future work, where finer scale flow structures are expected and some kind of non-Newtonian rheology or turbulence closure will have to be used.
- Both compared codes were validated for the case of Poiseuille flow of Newtonian fluid where the analytical solution is known. Both codes (finite-difference and finite-volume) resolved very well the expected known solution and there were no significant differences observed for numerical solutions obtained on comparable grids. This agreement on a simple channel case is taken as a starting point for the investigation of the behavior of both numerical codes on more complex geometries, where the analytical reference solutions are not available.
- First comparison of numerical simulations and their results have shown that the immersed boundary FDM is quite robust, fast and easy to modify for various geometries. On the other hand the computations using our older code based on FVM are more time consuming, more sensitive to grid structure and distortion.
- The simplicity of the newly developed immersed boundary finite-difference code outweighs its possible accuracy degradation in certain geometries. This code is primarily aimed at testing various artificial boundary conditions at multiply branched channels. For such purpose the code proved to be a perspective and valuable tool.

The results presented in this study are still more qualitative than quantitative. This is the direction where our future investigation will focus before some new experimental boundary conditions and mathematical models will be implemented in the code.

Acknowledgment

The financial support for the present work was partly provided by the *Czech Technical University, Faculty of mechanical engineering* under the *Grant No. SGS19/154/OHK2/3T/12* and partly by the European Regional Development Fund-Project “Center for Advanced Applied Science” No.CZ.02.1.01/0.0/0.0/16 019/0000778 .

References

- [1] J.D. Anderson: *Computational Fluid Dynamics - The Basics with Applications*. McGraw-Hill, 1995.
- [2] L. Beneš, P. Louda, R. Keslerová, K. Kozel, and J. Štigler: Numerical simulations of flow through channels with T-junction. *Applied Mathematics and Computation*, 219(13), 2013.
- [3] J.S. Bramley and D.M. Sloan: Numerical solution for two-dimensional flow in a branching channel using boundary-fitted coordinates. *Computers & Fluids*, 15(3):297–311, 1987.
- [4] A.J. Chorin: A numerical method for solving incompressible viscous flows problems. *Journal of Computational Physics*, 2(1):12–26, 1967.
- [5] A.J. Chorin: The numerical solution of the Navier-Stokes equations for an incompressible fluid. *Bulletin of the American Mathematical Society*, 73:928–931, 1967.
- [6] C.A.J. Fletcher: *Computational Techniques for Fluid Dynamics*, volume 1-2 of *Springer Series in Computational Physics*. Springer-Verlag Berlin Heidelberg, 2 edition, 1991.
- [7] C. Hirsch: *Numerical computation of internal and external flows*, volume 1,2. John Wiley & Sons, 1988.
- [8] M.Y. Kang, J. Hwang, and J.W. Lee: Effect of geometric variations on pressure loss for a model bifurcation of the human lung airway. *Journal of Biomechanics*, 44(6):1196–1199, 2011.
- [9] R. Keslerová and D. Trdlička: Numerical solution of viscous and viscoelastic fluids flow through the branching channel by finite volume scheme. *Journal of Physics: Conference Series*, 633:012128, 2015.
- [10] K. Kozel and R. Keslerová: Numerical study of viscous and viscoelastic fluids flow. *Journal of Math-for-Industry*, 3(3), 2011.
- [11] A. Lancmanová: Comparison of numerical methods for unsteady simulations of incompressible fluids flows. Master's thesis, Czech Technical University in Prague, 2020. (in Czech).
- [12] A. Lancmanová and T. Bodnár: Steady incompressible flow through a branched channel. In T. Bodnár, T. Neustupa, and D. Šimurda, editors, *Proceedings Topical Problems of Fluid Mechanics 2021*, pages 87–94. Institute of Thermomechanics CAS, 2021.
- [13] Y. Liu, R.M.C. So, and C.H. Zhang: Modeling the bifurcating flow in an asymmetric human lung airway. *Journal of Biomechanics*, 36(7):951–959, 2003.
- [14] B.J. Medhi, V. Agrawal, and A. Singh: Experimental investigation of particle migration in suspension flow through bifurcating microchannels. *AIChE Journal*, 64(6):2293–2307, 2018.
- [15] K. Pradhan and A. Guha: Fluid dynamics of a bifurcation. *International Journal of Heat and Fluid Flow*, 80:108483, 2019.
- [16] M.M. Reddy and A. Singh: Flow of concentrated suspension through oblique bifurcating channels. *AIChE Journal*, 60(7):2692–2704, 2014.
- [17] Y. Shang, J. Dong, L. Tian, K. Inthavong, and J. Tu: Detailed computational analysis of flow dynamics in an extended respiratory airway model. *Clinical Biomechanics*, 61:105–111, 2019.
- [18] E. Turkel: Preconditioned methods for solving the incompressible and low speed compressible equations. *Journal of Computational Physics*, 72(2):277–298, 1987.