ENCIT 2018

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0357
BLOOD FLOW DYMANICS SIMULATION IN CORONARY ARTERY WITH DRUG-ELUTING STENT USING FINITE ELEMENT METHOD

Leandro Marques
Gustavo Anjos
Jose Pontes
State University of Rio de Janeiro - UERJ, R. Fonseca Teles 524, 20550-013, Rio de Janeiro, Brazil
marquesleandro67@gmail.com,gustavo.anjos@uerj.br, jose.pontes@uerj.br

Abstract: *The present work aims at developing a computational framework to simulate coronary artery flows in cartesian coordinates. The Finite Element Method (FEM) is used to solve the governing equations of the blood flow in coronary artery with drug-eluting stent placed. The blood was modeled as single-phase, incompressible and newtonian fluid. The Navier-Stokes equation is shown according to the stream-vorticity formulation with coupled species transport equation. The Taylor-Galerkin scheme were used to decrease spurious oscillations as seen for moderate to high Reynolds number.*

Keywords: *Streamfunction-Vorticity Formulation, Finite Element Method, Taylor-Galerkin Method, Drug-Eluting Stent, Hemodynamics.*

1. INTRODUCTION

According to the World Health Organization (2017), more people die annually from the cardiovascular diseases (CVDs) that from any other cause in the world. An estimated 17.7 million people died from CVDs in 2015, representing 31% of all global deaths. About 41% of these deaths were due to coronary artery disease (CAD). The leading cause of the CAD is atherosclerosis where the diameter of the vessel is decreased. Two treatments can be performed: coronary artery bypass grafting (CABG) or percutaneous transluminal coronary angioplasty (PTCA). The PTCA is a minimally invasive procedure where a small wire tube, called stents, is placed. This work aims to develop a Finite Element code for stream-vorticity formulation coupled species transport equation and to know how the dynamics of blood flow in coronary artery with atherosclerosis and with stents struts placed.

The dynamics of blood flow in coronary artery and possible influence of stents struts with computational fluid dynamics (CFD) requires a robust numerical method to compute the solution of the differential equations in a relevant model. The equations that govern the dynamics of blood flow in a coronary artery were developed according to continuum media assumption. Thus, the universal conservation laws such as conservation of mass, conservation of momentum and conservation of species transport were used. The blood was modeled as single-phase, incompressible and newtonian fluid, the diffusion coefficient was considered as constant. The Navier-Stokes equation is shown according to the 2D stream-vorticity formulation with coupled species transport equation in a Finite Element Method approach.

The domain was discretized on an unstructured triangular mesh using the *GMSH* open source as proposed by Geuzaine and Remacle (2009). According to decoupling between velocity field and pressure field achieved by stream-vorticity formulation, the linear triangular element was used. The equations were discretized in time by Taylor series expansion remaining the second order terms to decrease spurious oscillations as seen for moderate to high Reynolds number. Then, the Galerkin formulation was used to discretize in space. Therefore, the Taylor-Galerkin scheme was used as proposed by Donea (1984).

The computational development was done in *Python* language using object-oriented programming paradigm with the aim of reusability and further development. The code validation was made by comparison numerical solution and analytical solution of the *Poiseuille flow*. The comparison of velocity field was done for lid-driven cavity flow with those shown by Ghia *et al.* (1982) and Marchi *et al.* (2009). The dynamics of blood flow and species transport in coronary artery was investigated in 2 test cases as suggered by Wang *et al.* (2017), however modified for 2D cartesian coordinates. The simulation was shown using *Paraview* open source as proposed by Henderson (2007).

2. MATHEMATICAL MODEL

A 2-dimensional Finite Element Method approach is employed to analyse the dynamics of blood flow in coronary artery with atherosclerosis and possible influence of stents struts. The governing equations were developed according to continuum media assumption. Thus, the universal conservation laws such as conservation of mass, conservation of momentum and conservation of species transport were used. The blood was modeled as single-phase, incompressible and newtonian fluid, the diffusion coefficiente was considered as constant. The Navier-Stokes equation is shown according to stream-vorticity formulation with coupled species transport equation.

$$\frac{\partial \omega}{\partial t} + \mathbf{v}.\nabla\omega = \frac{1}{Re}\nabla^2\omega \tag{1}$$

$$\nabla^2\psi = -\omega \tag{2}$$

$$\mathbf{v} = \mathbf{D}\psi \tag{3}$$

$$\frac{\partial c}{\partial t} + \mathbf{v}.\nabla c = \frac{1}{ReSc}\nabla^2 c \tag{4}$$

Where, ω is the vorticity field, ψ is the stream function field, *c* is the concentration field, $\mathbf{v} = (u, v)$ is the velocity field, $\mathbf{D}$ = $[\partial/\partial y, -\partial/\partial x]$ is a mathematical operator, $Re = \rho u D/\mu$ is the Reynolds number, $Sc = \nu/D$ is the Schmidt number, *x* and *y* are the independent spatial variables and *t* is the time variable.

2.1 Finite Element Method

The domain was discretized on an unstructured triangular mesh using the *GMSH* open source as proposed by Geuzaine and Remacle (2009). According to decoupling between velocity field and pressure field achieved by stream-vorticity formulation, the use of linear triangular element was used. The equations were discretized in time by Taylor series expansion remaining the second order terms to decrease spurious oscillations as seen for moderate to high Reynolds number. Then, the Galerkin formulation was used to discretize in space. Therefore, the Taylor-Galerkin scheme was used as proposed by Donea (1984). The governing equations in matrix form used in this paper were:

$$\begin{aligned}\left[\frac{M}{\Delta t} + \frac{1}{Re}\Big[K_{xx} + K_{yy}\Big]\right]\omega^{n+1} = \frac{M}{\Delta t}\omega^n - u\cdot G_x\omega^n - v\cdot G_y\omega^n \\ - u\frac{\Delta t}{2}\big[uK_{xx} + vK_{yx}\big]\omega^n - v\frac{\Delta t}{2}\Big[uK_{xy} + vK_{yy}\Big]\omega^n\end{aligned} \tag{5}$$

$$\Big[K_{xx} + K_{yy}\Big]\psi = M\omega \tag{6}$$

$$Mu = G_y\psi \tag{7}$$

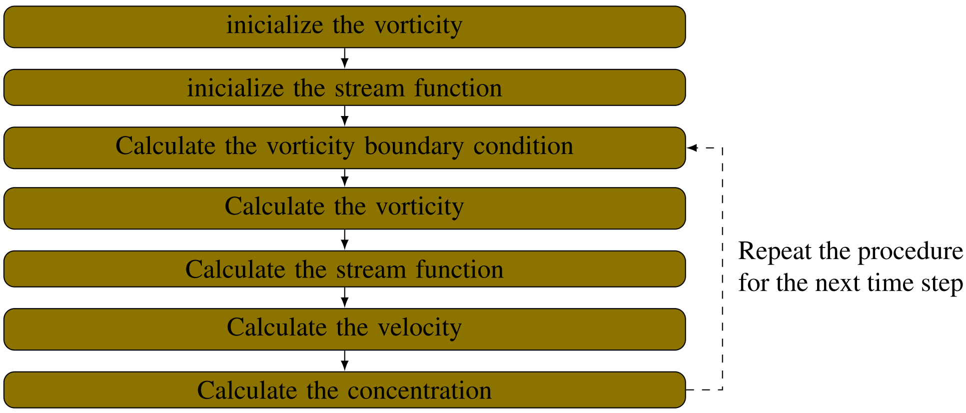
$$Mv = -G_x\psi \tag{8}$$

$$\begin{aligned}\left[\frac{M}{\Delta t} + \frac{1}{ReSc}\Big[K_{xx} + K_{yy}\Big]\right]c^{n+1} = \frac{M}{\Delta t}c^n - u\cdot G_xc^n - v\cdot G_yc^n \\ - u\frac{\Delta t}{2}\big[uK_{xx} + vK_{yx}\big]c^n - v\frac{\Delta t}{2}\Big[uK_{xy} + vK_{yy}\Big]c^n\end{aligned} \tag{9}$$

Where, *M* is mass matrix, G_x and G_y are gradient matrix, K_{xx}, K_{xy}, K_{yx} and K_{yy} are stiffness matrix. The last term of the Eqs. 5 and 9 is known as numerical diffusion and it decrease the spurious oscillations as seen for moderate to high Reynolds numbers. For scalars, *Taylor Galerkin Method* and *Characteristic Galerkin* produce the same result as showed by Lohner *et al.* (1984). The superscripts *n+1* and *n* are the scalar that will be calculated and that was calculated in the previous time step, respectively.

2.2 Numerical Solution

The computational development was done in *Python* language using object-oriented programming paradigm with the aim of reusability and further development. The linear system of equations that come from implementing the FEM is solved throught iterative method *Conjugate Gradient Solver* available in the public library for scientific tools *SciPy* maintained by Jones *et al.* (2001). The solution algorithm used is shown below:



The first and second steps are out of time loop, while the third to the seventh step are inside of time loop. The application of the boundary condition in the equation can be before loop, except for the vorticity equation (*fourth step*) that the boundary condition must be applied at each time step.

3. VALIDATION

The code validation was made by comparison numerical solution and analytical solution of the *Poiseuille flow*. The comparison of velocity field was done for lid-driven cavity flow with those shown by Ghia *et al.* (1982) and Marchi *et al.* (2009).

3.1 Poiseuille Flow

A single-phase flow, steady and fully developed of an incompressible and newtonian fluid between parallel horizontal plates and stationary is maintained due to a pressure gradient. This flow is known as *Poiseuille flow*. In Fig. 1 is shown a schematic representation of the numerical domain used to simulate the Poiseuille Flow problem, where no-slip condition were used at the top and bottom walls, while an inflow and outflow conditions were set in the plane $x = 0$ and $x = 5L$ respectively.

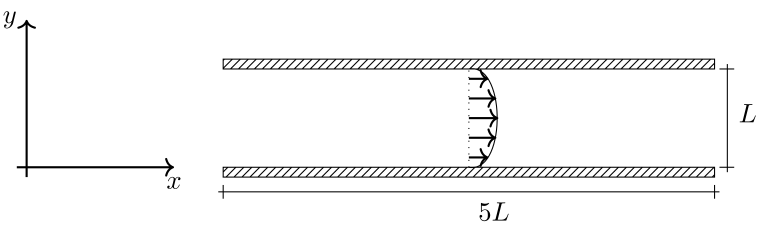


Figure 1: Poiseuille flow

The boundaries conditions used were:

- *inflow condition*: the normal velocity component is set to null value $v = 0$. The tangent velocity component is set to parabolic profile by exact solution, that is, $u = \left[4u_{max}y/L^2\right]\left[L - y\right]$, where $u_{max} = 1.5$. The streamfunction is also specified and its value is defined according to continuity equation for an incompressible fluid. Thus, its value is $\psi = \left[2u_{max}y^2/3L^2\right]\left[3L - 2y\right]$.
- *No-slip condition*: all the velocity components are specified with null value $u = 0$ and $v = 0$. The streamfunction is also specified $\psi = 1$ in top plate and $\psi = 0$ in bottom plate.

- *outflow condition*: no value is specified. The derivatives of the tangent velocity component, of the normal velocity component and of streamfunction are set to null values, that is, $\partial u/\partial n = 0$, $\partial v/\partial n = 0$ and $\partial \psi/\partial n = 0$ respectively.

The velocity field profile is calculated by equation below:

$$u = \frac{4u_{max}}{L^2} y \left[L - y\right] \tag{10}$$

Where, u_{max} is maximum velocity and its value is $u_{max} = 1.5$, L is non-dimensional width between parallel plates and its value is $L = 1$ and y is length between parallel plates and ranges $y = [0, 1]$. The domain was discretized using a linear triangular mesh with 3835 nodes and 7299 elements.

In Fig. 2 is shown the evolution of velocity field profile in time when $Re = 100$, the comparasion is also shown between numerical solution and analytical solution for steady state. As can be seen, the code showed a satisfactory result.

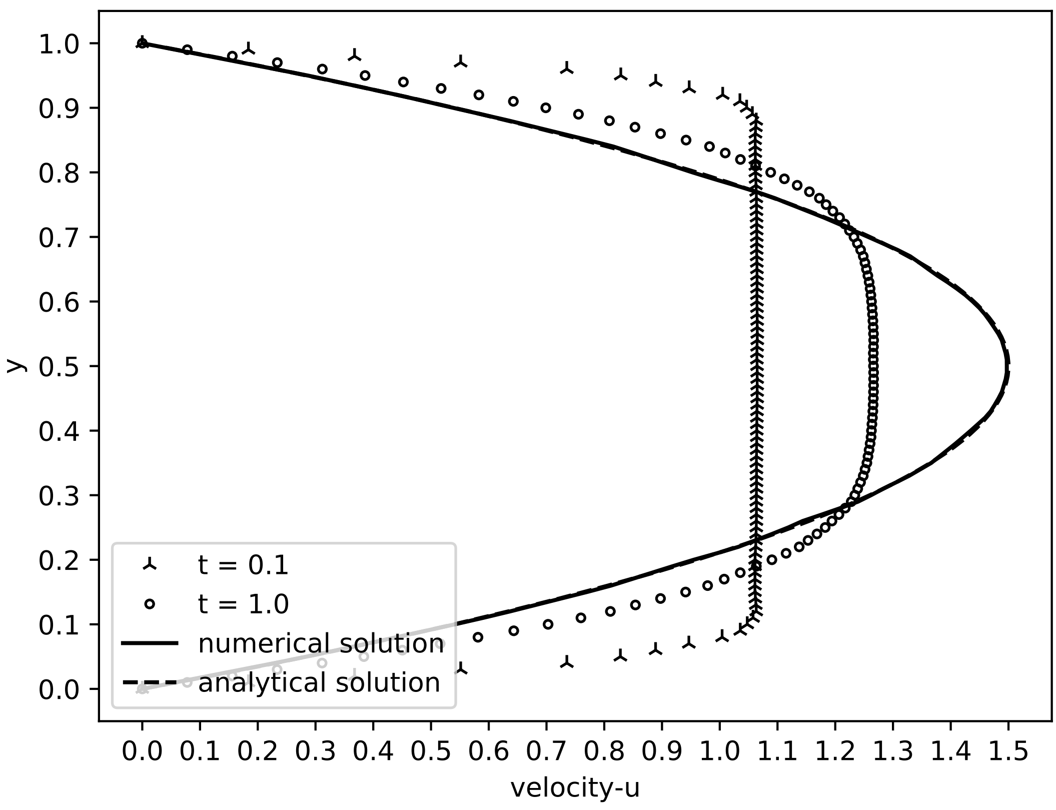


Figure 2: Evolution of velocity field profile in time for $Re = 100$ and the comparison between numerical solution and analytical solution.

3.2 Lid-driven Cavity Flow

A flow on cavity when side and bottom plates are stationary and top plate moves with velocity constant such that $U = top = 1$ is known as *lid-driven cavity flow*. In Fig. 3 is shown a schematic representation of the numerical domain used to simulate the Lid-driven Cavity Flow problem, where no-slip condition were used at the bottom and side walls, while at top wall, the velocity was set $u = U_{top}$.

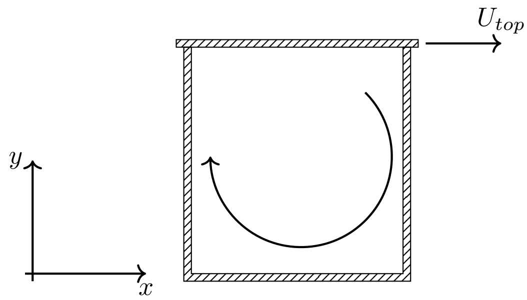


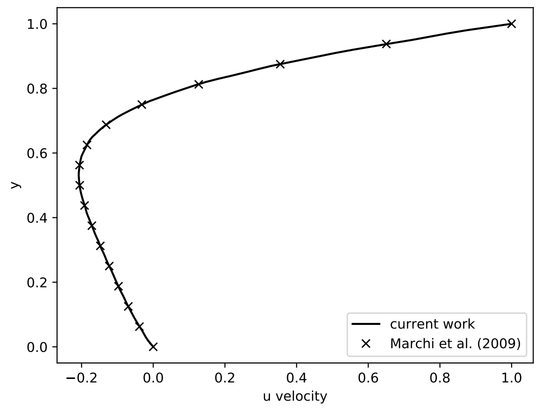
Figure 3: Lid-driven cavity flow

Were simulated flows for the following Reynolds numbers: (*Re*): 10, 100, 400 and 1000. The dimensions domain in x-direction and y-direction are [0,1]. We used a mesh with 1563 nodes and 2988 elements

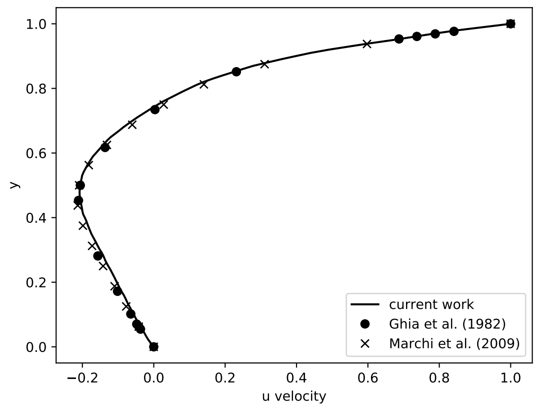
The boundaries conditions used were:

- *top plate moves*: all the velocity components are specified with $v = 0$ and $u = U_{top}$, where $U_{top} = 1$. The streamfunction is also specified and its value is $\psi = 0$.
- *No-slip condition*: This condition is used on side and bottom plates. All the velocity components are also specified with null value $u = 0$ and $v = 0$. The streamfunction is specified as $\psi = 0$.

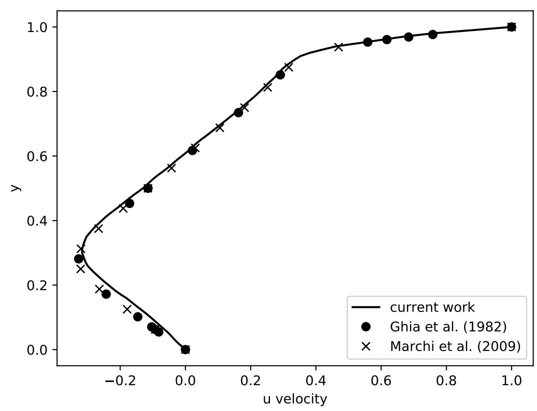
The Figs. 4 and 5 shown profile of u and v components respectvely for several Reynolds numbers. The results were compared with Ghia *et al.* (1982) and Marchi *et al.* (2009). Therefore, the code shown a satisfactory result.



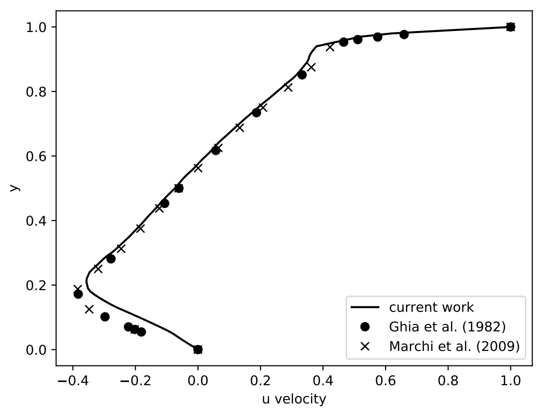
(a)



(b)

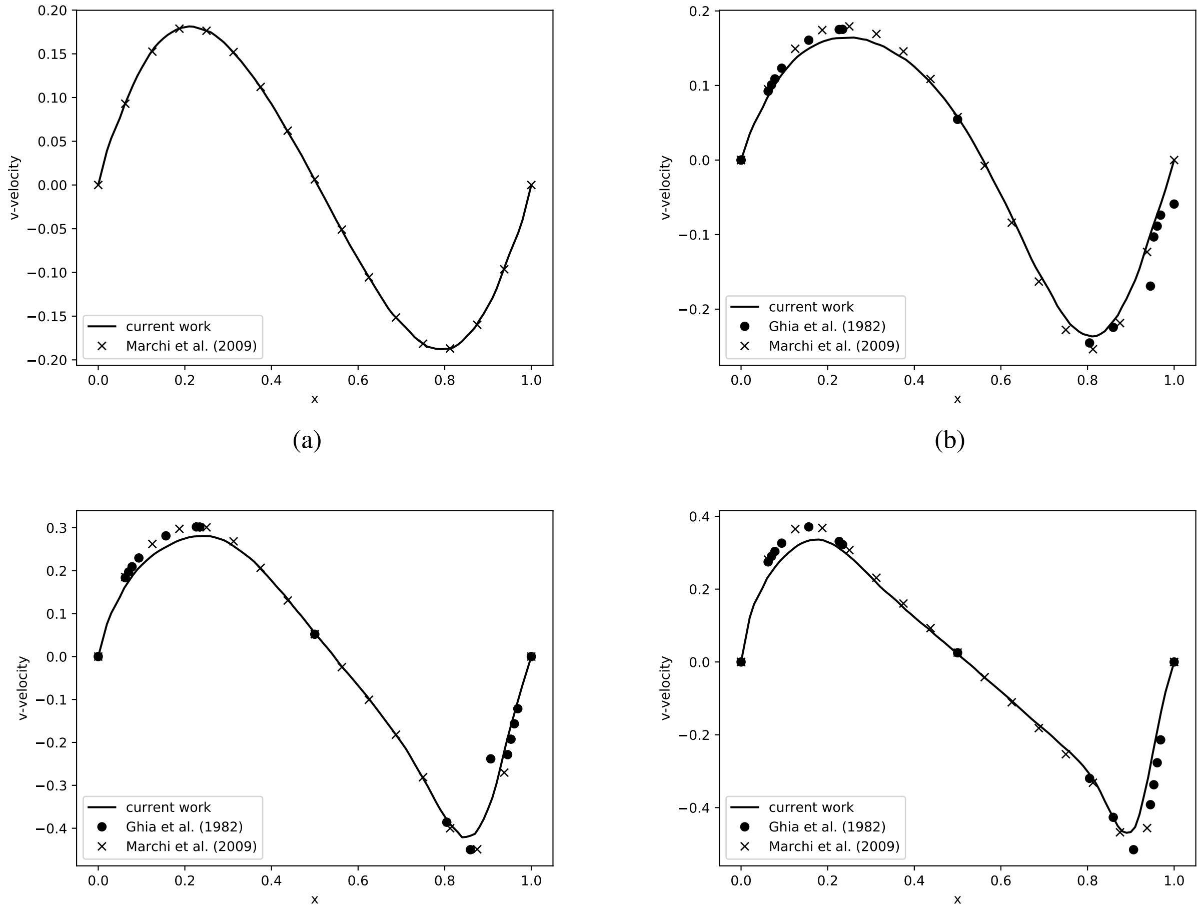


(c)



(d)

Figure 4: Centerline u velocity profile ($x = 0.5$) in a lid-driven cavity for different Reynolds numbers: (a) 10 (b) 100 (d) 400 (f) 1000.



(a) (b)

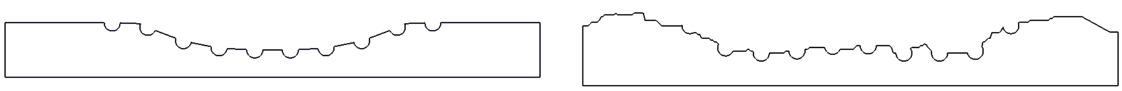
(c) (d)

Figure 5: Centerline v velocity profile ($y = 0.5$) in a lid-driven cavity for different Reynolds numbers: (a) 10 (b) 100 (d) 400 (f) 1000.

4. RESULTS AND DISCUSSION

Some results of simulations are shown to demonstrate its capability of using unstructured triangular meshes on various geometries and combination of geometries. Numerical results are given for several cases of blood flows in artery when $Sc = 10$. The post-processing was performed by open source software *PARAVIEW* proposed by Henderson (2007). The lumen radius of a typical artery is about $R = 0.0015$m, viscosity in the lumen are set to $\mu = 0.0035$Pa.s. and density $\rho = 1060$kg/m^3 as suggested by Bozsak *et al.* (2014). According to Kessler *et al.* (1998), the velocity of the flow at coronary artery is $v = 12$cm/s. Therefore, the Reynolds number is $Re = 54.5$.

The Navier-Stokes equation is used according to stream-vorticity formulation with species transport equation coupled for 2 different geometries proposed by Wang *et al.* (2017) and is shown in Fig. 6, however modified for cartesian coordinates. According to symmetry on y coordinate, half domain was simulated.



(a) (b)

Figure 6: Non-dimensional geometry for blood flow dynamics in coronary arteries. The channel length L = 10R is based on the channel width R = 1. (a) Curved Channel with Stent and (b) Real Channel with Stent.

The boundaries conditions used were:

- *inflow condition*: the normal velocity component is set to null value $v = 0$. The tangent velocity component is set to parabolic profile by exact solution, that is, $u = u_{max}\left[1 - \left(y/L\right)^2\right]$, where $u_{max} = 1.5$. The streamfunction is also specified and its value is defined according to continuity equation for an incompressible fluid. Thus, its value is $\psi = \left[u_{max}y/3\right]\left[3 - \left(y/L\right)^2\right]$.

- *No-slip condition*: this condition is used on bottom plate. all the velocity components are specified with null value $u = 0$ and $v = 0$. The streamfunction is also specified $\psi = 1$ and the derivative of concentration has null value $\partial c/\partial n = 0$.
- *outflow condition*: no value is specified. The derivatives of the tangent velocity component, of the normal velocity component and of streamfunction are set to null values, that is, $\partial u/\partial n = 0$, $\partial v/\partial n = 0$, $\partial \psi/\partial n = 0$ and $\partial c/\partial n = 0$ respectively.
- *Free-slip condition*: used when a symmetry condition is desired. The normal velocity component is set to null value $v = 0$ as well as the streamfunction $\psi = 0$. The derivative of the tangent velocity component and the derivative of the concentration are also set to null value $\partial u/\partial n = 0$ and $\partial c/\partial n = 0$ respectively.
- *Strut condition*: used on the stent. The normal velocity component and the tangent velocity component are specified with null value $u = 0$ and $v = 0$. The streamfunction and the concentration are also specified $\psi = 1$ and $c = 1$ respectively.

4.1 Curved Channel with Stent

For the case when coronary artery has atherosclerosis and drug-eluting stent is placed, the problem is modeled as a parallel and curved plates flow and the stent is modeled by 10 semi-circles uniformely spaced. The geometry used promotes a smooth reduction of length between the bottom and top plates. Were considered 40% of channel obstruction due to atherosclerosis. The Fig. 10 shown velocity field profile along y coordinates in centerline ($x = 5R$).

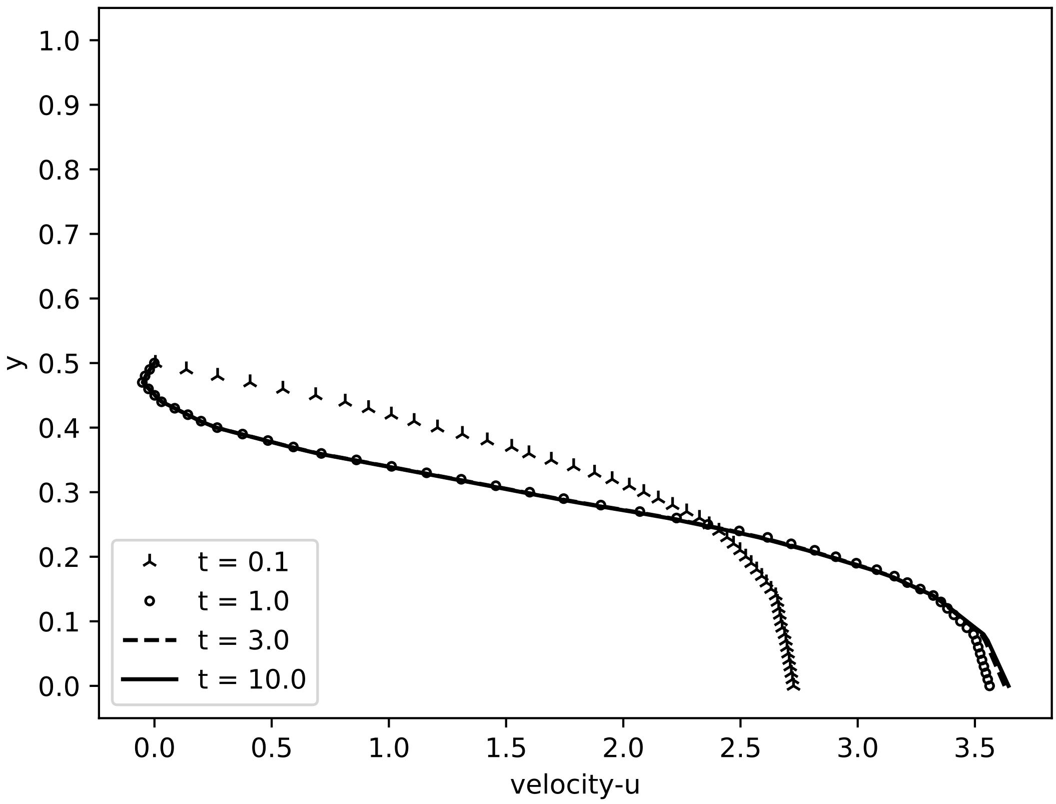


Figure 7: Evolution of velocity profile in time for Curved Channel with Drug-eluting Stent.

In Figs 8 and 9, are shown the velocity and concentration evolution in time and space for half domain according to symmetry y coordinate. The concentration field is represented with non-dimensional values when the red color is $c = 1$ and blue color $c = 0$.

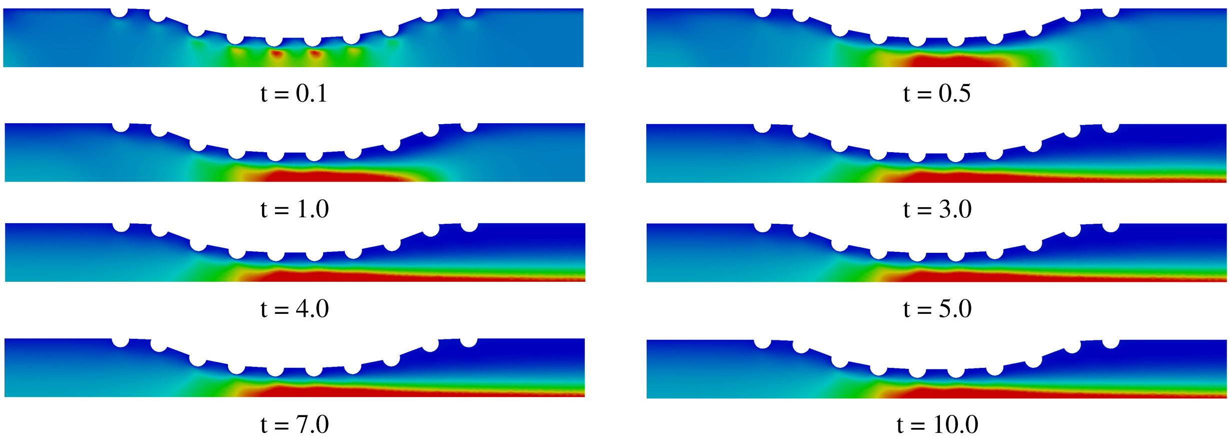


Figure 8: Evolution in time and space of velocity field for Curved Channel with Drug-eluting Stent.

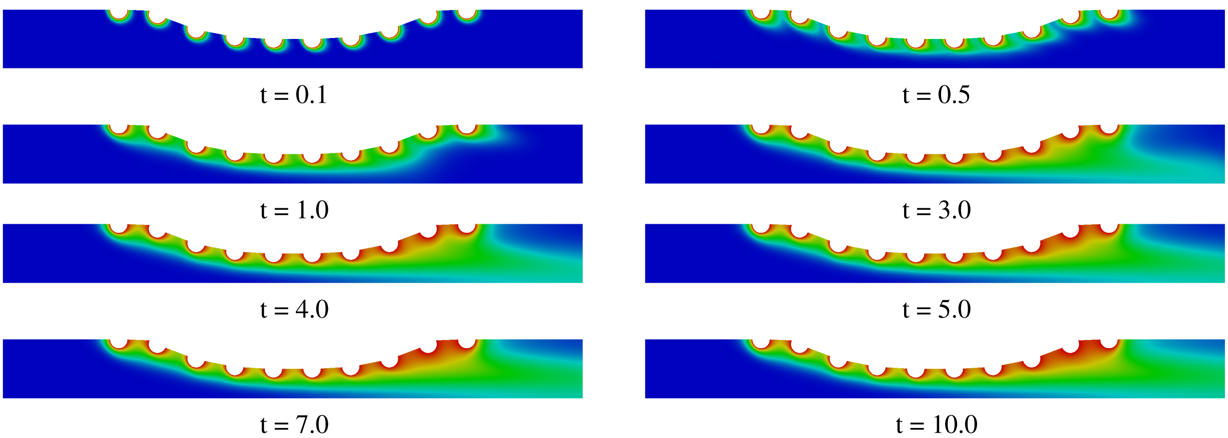


Figure 9: Evolution in time and space of concentration field for Curved Channel with Drug-eluting Stent.

4.2 Real Channel with Stent

In this case, the real coronary artery with atherosclerosis and drug-eluting stent placed is performed. The geometry was taken using image processing from a real coronary artery photography. The stent is modeled by 10 semi-circles uniformely spaced. As in other case, the geometry used promotes a reduction of length between the bottom and top plates when were considered 40% of channel obstruction due to atherosclerosis. The Fig. 10 shown velocity field profile along y coordinates in centerline $(x = 5R)$.

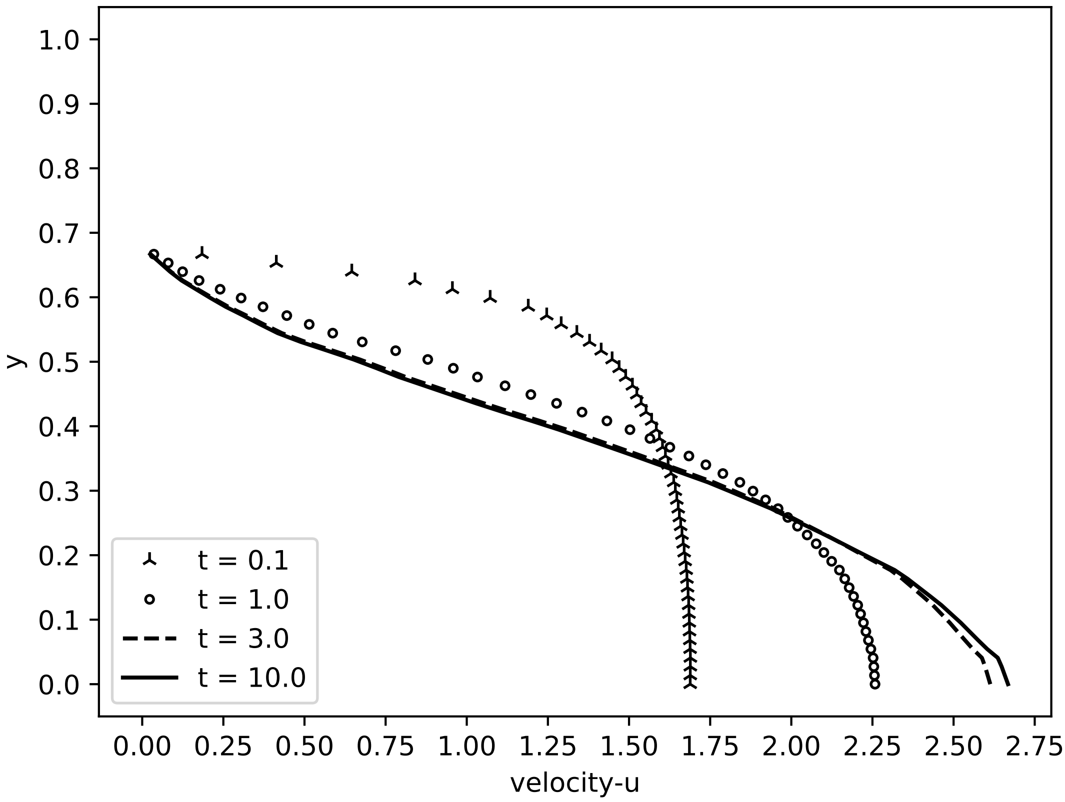


Figure 10: Evolution of velocity profile in time for Curved Channel with Drug-eluting Stent.

In Figs 11 and 12, are shown the velocity and concentration evolution in time and space for half domain according to symmetry y coordinate. The concentration field is represented with non-dimensional values when the red color is $c = 1$ and blue color $c = 0$.

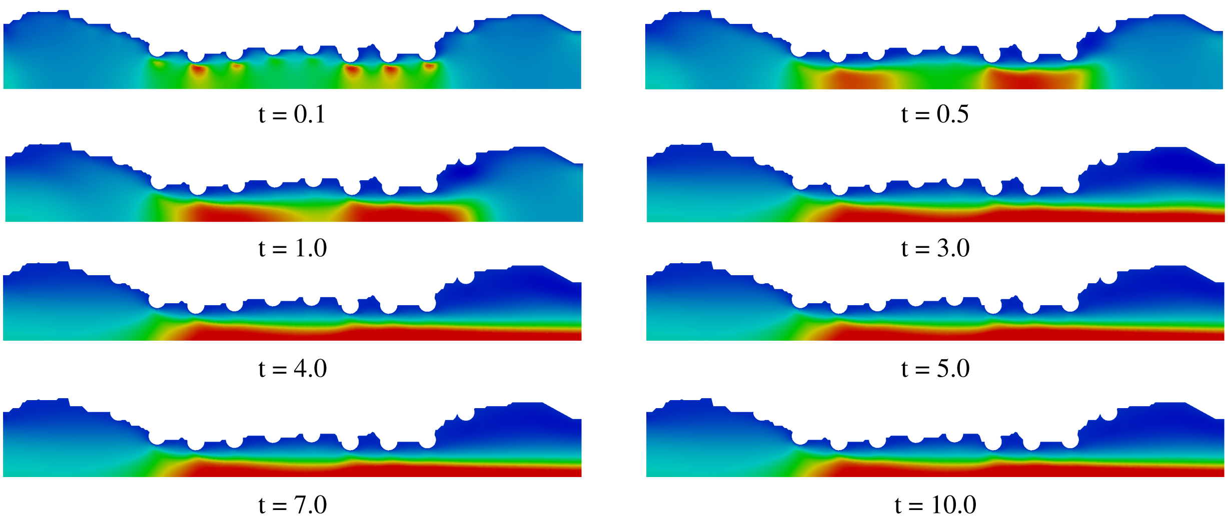


Figure 11: Evolution in time and space of velocity field for Real Channel with Drug-eluting Stent.

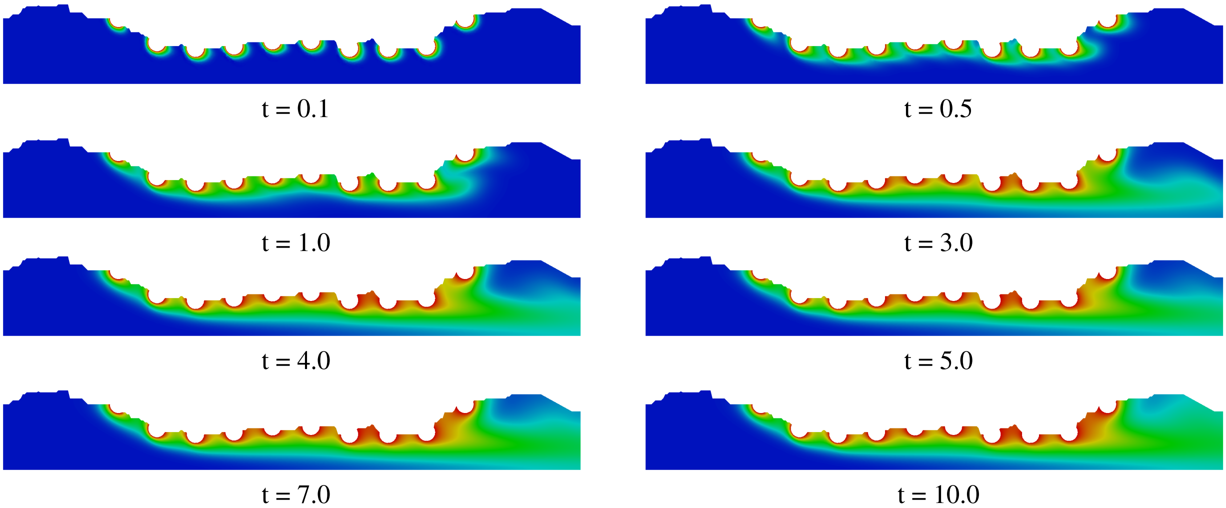


Figure 12: Evolution in time and space of concentration field for Real Channel with Drug-eluting Stent.

5. CONCLUSION

In this work, a numerial code for Navier-Stokes equation according to stream-vorticity formulation with species transport equation coupled was developed using Finite Element Method. The Taylor-Galerkin scheme was applied to decrease the spurious oscillations as seen for moderate to high Reynolds number. The code proved to be effective by results presented in validation cases. The dynamics of blood flow was shown to a coronary artery with atherosclerosis and drug-eluting stent placed. Therefore, the streamfunction and vorticity formulation showed an useful approximation for to calculate the velocity and concentration fields since the variables are scalars allowing then a smooth implemention.

6. ACKNOWLEDGEMENTS

The authors thank the FAPERJ (Research Support Foundation of the State of Rio de Janeiro) for its financial support.

7. REFERENCES

Bozsak, F., J-M., C. and Barakat, A., 2014. "Modeling the transport of drugs eluted from stents: physical phenomena driving drug distribution in the arterial wall". *Biomech Model Mechanobiol*, Vol. 13, pp. 327–347. doi:10.1007/s10237-013-0546-4.

Donea, J., 1984. "A taylor-galerkin method for convective transporte problems". *International Journal for Numerical Methods in Engineering*.

Geuzaine, C. and Remacle, J., 2009. "Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities". *International Journal for Numerical Methods in Engineering*.

Ghia, U., Ghia, K.N. and Shin, C.T., 1982. "High-re solutions for incompressible flow using the navier-stokesequations and a multi-grid." *Journal of Computational Physics*.

Henderson, A., 2007. "Paraview guide, a parallel visualization application". *Kitware Inc.*

Jones, E., Oliphant, T. and Peterson, P., 2001. "Scipy - open source scientific tools for python". URL `http://www.scipy.org/`. [Online; accessed 15/03/2018 09:14].

Kessler, W., Moshage, W., Galland, A., Zink, D., Achenbach, S., Nitz, W., Laub, G. and Bachmann, K., 1998. "Assessment of coronary blood flow in humans using phase difference mr imaging comparison with intracoronary doppler flow measurement." *International Journal of Cardiac Imaging*.

Lohner, R., Morgan, K. and Zienkiewicz, O.C., 1984. "The solution of non-linear hyperbolic equation systems by the finite element method". *International Journal of Numerical Methods in Fluids*.

Marchi, C.H., Suero, R. and Araki, L.K., 2009. "The lid-driven square cavity flow: Numerical solution with a 1024 x 1024 grid." *Journal of the Brazilian Society of Mechanical Sciences and Engineering*.

Wang, H., McGinty, S., Lucena, R., Pontes, J., Anjos, G. and Mangiavacchi, N., 2017. "Dynamics of blood flow in coronary artery". *International Congress of Mechanical Engineering*.

World Health Organization, W., 2017. "Cardiovascular diseases". URL `"www.who.int/cardiovascular_diseases/en/"`. [Online; accessed 03/05/2018 11:55].

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.