Blood flow in artificial bypass graft: a numerical study

L. C. Sousa, C. F. Castro & C. C. António  
*IDMEC - FEUP, University of Porto, PORTO, Portugal*

B. Relvas  
*IDMEC - Pólo FEUP, PORTO, Portugal*

ABSTRACT: The purpose of this study is to obtain a first insight of blood physics behaviour through a bypass graft. The paper describes a numerical analysis by the finite element method of steady blood flow through an idealized bypass graft around a stenosed artery. In this study the biochemical and mechanical interactions between blood and vascular tissue are neglected and no-slip boundary conditions including the graft are considered. Results are visualized for a better understanding of the flow characteristics such as distributions of the flow pattern, stagnation flow and recirculation zones. The outcomes will contribute to characterize the physiology of bypass grafts in the human circulatory system.

1 MOTIVATION

Arterial diseases such as wall conditions may cause blood flow disturbances leading to clinical complications in areas of complex flow like in coronary and carotid bifurcations or stenosed arteries. It is well established that once a mild stenosis is formed in the artery, biomechanical parameters resulting from the blood flow and stress distribution in the arterial wall contribute to further progression of the disease (Deplano & Siouffi 1999). Although blood flow is normally laminar, the periodic unsteadiness or pulsatile nature of the flow makes possible the transition to turbulence when the artery diameter decreases and velocities increase.

A detailed understanding of local hemodynamic environment, influence of wall modifications on flow patterns and long-term adaptations of the vascular wall can have useful clinical applications, especially in view of reconstruction and revascularization operations helping doctors or surgeons to make a diagnosis or to plan a surgery.

Nowadays, the use of computational techniques in fluid dynamics in the study of physiological flows involving blood is an area of intensive research (Taylor et al. 1998, Quarteroni et al. 2003). Flow visualization techniques and non-invasive medical imaging data acquisition such as computed tomography, angiography or magnetic resonance imaging, make feasible to construct three dimensional models of blood vessels. Measuring techniques such as Doppler ultrasound have improved to provide accurate information on the flow fields.

In medical practice bypass grafts are commonly used as an alternative route around strongly stenosed or occluded arteries. For inflow operations artificial grafts are considered (Abraham et al. 2004, Huang et al. 1995, Su et al. 2005). They perform well when the arterial flow is high and it has been shown over the years that they provide durable results. The main goal of this work is to obtain a first insight into the simulation flow through an idealized bypass graft using a developed computer program based on the finite element method. Modelling the fluid flow in idealized geometries will help to investigate the issues related to bypass anastomosis as knowing the blood flow, the velocity and stress fields it is possible to find a geometry that leads to small gradient hemodynamic flow and recirculation zones.

2 COMPUTATIONAL STRATEGY

In this current study, due to the complexity of the cardiovascular system, a preliminary analysis aiming suitable simplifying assumptions for the mathematical modelling process is needed.

The numerical analysis of the blood flow phenomena uses the finite element method approach and a geometrical model of the artery and artificial bypass graft. Blood flow is described by the incompressible Navier-Stokes equations and the simulation is carried out under steady flow conditions. Although blood flow in arteries is substantially influenced by unsteady flow phenomena steady state re-
results allow to gain an in-depth understanding of the fluid physics.

In diseased vessels which are often the subject of interest, the arteries are even less compliant and wall motion is further reduced and the assumption of zero wall motion is utilized in most approximations. In the present work, blood flow in an idealized bypass graft is studied considering boundary conditions similar to physiological circumstances. Biochemical and mechanical interactions between blood and vascular tissue are neglected.

2.1 Mathematical model

The fluid flow is governed by the equation of Navier-Stokes. This equation results from the application of the principle of mass conservation. A non-Newtonian viscosity model for simulating pulsatile flow in artery is adopted in this study. Considering blood flow an incompressible non-Newtonian flow and neglecting body forces, the equation of continuity and the Navier-Stokes equations become:

\[ \nabla \cdot \mathbf{v} = 0 \]

\[ \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = -\frac{1}{\rho} \nabla p + \mu \nabla^2 \mathbf{v} \]  

(1)

where \( \mathbf{v} \) is the velocity field, \( \mu \) the dynamic viscosity, \( \rho \) the blood density and \( p \) the pressure. Considering the pseudo-constitutive relation for the incompressibility constraint the continuity equation is replaced by (Babuska 1973, Babuska et al. 1980):

\[ p = -\frac{1}{\varepsilon} \nabla \cdot \mathbf{v} \]  

(2)

Where \( \varepsilon \) is the penalty parameter generally assigned to \( 10^{-8} \) or \( 10^{-9} \). The first set of Equation (1) is eliminated and the Navier-Stokes equations become:

\[ \rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\frac{1}{\varepsilon} \nabla (\nabla \cdot \mathbf{v}) + \mu \nabla^2 \mathbf{v} \]  

(3)

Under such conditions the pressure is eliminated as a field variable since it can be recovered by the approximation of Equation (2). If the standard Galerkin formulation is applied it is necessary to use compatible spaces for the velocity and the pressure in order to satisfy the Babuska-Brezzi stability (Babuska 1973, Babuska et al. 1980, Sousa et al. 2002). Unfortunately, not every combination of interpolation functions for pressure and velocity works, as they are required to satisfy the Babuska-Brezzi conditions or pass the mixed patch test in the incompressible limit. In this work reduced integration is used for the terms related with pressure in order to avoid locking effects and to obtain a stabilized finite element solution. A smoothing technique is applied to get continuous fields for pressure and deviatoric stresses.

In blood flow high Reynolds numbers appear and loss of unicity of solution, hydrodynamical instabilities and turbulence are caused by the convective term in Equation (3). If Navier-Stokes equation is solved numerically by the Galerkin method, an unstable and oscillating solution is observed at higher speeds. A severely refined mesh can be used to avoid this phenomenon, but computation time might then become unacceptable.

The numerical scheme requires a stabilization technique in order to avoid oscillations in the numerical solution. In this study the streamline upwind Petrov-Galerkin method is applied in order to avoid the loss of accuracy (Brooks & Hughes 1982). This method is applied using modified velocity shape functions, \( W_i \):

\[ W_i = N_i + K_{SUPG} \frac{v \nabla N_i}{||v||} \]  

(4)

where \( N_i \) are the Galerkin shape functions and \( K_{SUPG} \) denotes the upwind parameter that controls the factor of upwind weighting. The resulting system of nonlinear equations is characterized by a non-symmetric matrix, and a special solver is adopted in order to reduce the bandwidth and the storage of the sparse system matrix; the skyline method is used in addition to some improvement of the Gauss elimination.

In this work a steady blood flow is considered and the first term of Equation (3) disappears becoming:

\[ \rho \varepsilon \nabla \cdot \mathbf{v} = -\frac{1}{\varepsilon} \nabla (\nabla \cdot \mathbf{v}) + \mu \nabla^2 \mathbf{v} \]  

(5)

Non-Newtonian property of blood is important in the hemodynamic effect and plays a significant role in vascular biology and pathology. In this study two non-Newtonian viscosity models are adopted. In the first, the viscosity is empirically obtained using Casson law for the shear stress relation (Perktold et al. 1991). Considering \( D_{II} \) the second invariant of the strain rate and \( c \) the red cell concentration, the shear stress \( \tau \) given by the generalized Casson relation is:

\[ \sqrt{\tau} = k_0 + k_1(c) \sqrt{2D_{II}} \]  

(6)

and the apparent dynamic viscosity \( \mu = \mu(c,D_{II}) \), a function of the red cell concentration,

\[ \mu = \frac{1}{2\sqrt{D_{II}}} \left( k_0 + k_1(c) \sqrt{2D_{II}} \right)^2 \]  

(7)

where parameters \( \mu_0 = 0.124 \text{ Ns}, k_0 = 0.6125 \) and \( k_1 = 0.174 \) were obtained fitting experimental data and considering \( c = 45\% \). For the second approach the Carreau-Yasuda constitutive model is adopted and dynamic viscosity is given by (Abraham et al. 2004):
\[
\mu(\dot{\gamma}) = \mu_\infty + \frac{\mu_0 - \mu_\infty}{1 + (\lambda \dot{\gamma})^b}^a
\]  

where \( \mu_0 = 0.116 \) Ns, \( \mu_\infty = 0.0035 \) Ns, \( a = 1.23 \), \( b = 0.64 \) and \( \lambda = 8.2 \) s. The shear rate, \( \dot{\gamma} \), is related to the second invariant of the strain rate tensor and can be directly obtained from the flow field as:

\[
\dot{\gamma} = \sqrt{2\varepsilon(v) : \dot{\varepsilon}(v)}
\]

where \( \dot{\varepsilon} \) is the strain rate tensor.

3 NUMERICAL RESULTS

3.1 Finite element approach

A graft is attached around an occlusion in the coronary artery, as an alternative route for blood flow. The first results presented here consider a 3D simulation. The boundary conditions for the flow field are parabolic inlet velocity, no-slip boundary conditions including graft and artery and a parallel flow condition at the outlet. Considering a 3D analysis the results are presented for Reynolds number equal to 50, defining the Reynolds number as \( Re = \frac{\rho v_{\text{max}} D}{\mu} \), where \( v_{\text{max}} \) is the maximum velocity at the inlet and \( D \) the artery diameter. Figure 1 shows the geometry and finite element mesh for the arterial bypass system analysed where the artery diameter is \( D = 10 \) mm and the graft diameter \( d = 7 \) mm. Due to symmetry only half of the bypass geometry is considered for the numerical simulation. The mesh consists of 11607 nodes and 9272 elements. Considering Casson constitutive model the velocity field is shown in Figure 2.

Figure 1. Geometry and finite element mesh for the arterial bypass system analysed.

Figure 2. Velocity field using Casson model, \( Re = 50 \), \( d = 7 \) mm.

Figures 3 and 4 show the longitudinal velocity and pressure contours on the plane of symmetry. It is observed that there are no recirculation zones as Reynolds number is not high; pressure increases as the flow exits the graft and the fluid pressure along the centerline is a constant function of axial distances. The 3D simulation is compared with a 2D one for the same geometry; simulated 2D velocity and pressure distributions presented in Figures 5 and 6 are similar to those obtained in the 3D simulation.

Figure 3. Plane of symmetry of 3D simulation. Velocity field (mm/s) using Casson model, \( Re = 50 \), \( d = 7 \) mm.

Figure 4. Plane of symmetry of 3D simulation. Pressure distribution \( (10^4 \text{ Pa}) \) using Casson model, \( Re = 50 \), \( d = 7 \) mm.

Figure 5. 2D simulation. Velocity field (mm/s) using Casson model, \( Re = 50 \), \( d = 7 \) mm.
Since 3D and 2D simulations present similar results the following discussion considering Reynolds number equal to 300 and comparing two viscosity models is based on 2D simulations. In Figures 7 and 8 the results are obtained using Casson’s law while Figures 9 and 10 correspond to the Carreau-Yasuda viscosity model. For these analyses two recirculation zones are observed one near the proximal entrance of the bypass graft and another near the toe of the distal anastomosis; it can also be deduced from Figures 7 and 9 that the size of these two regions is more extended when Carreau-Yasuda model is adopted.

### 3.2 Aspect ratio variation

Defining aspect ratio as the ratio of the graft diameter $d$ to artery diameter $D$, simulations for different aspect ratios are analysed in this work. Considering Carreau-Yasuda constitutive model the previous results given in Figures 9 and 10 corresponding to $d/D = 0.7$ are compared with new simulations for higher aspect ratio values. Velocity and pressure distributions for an aspect ratio of 0.8 are presented in Figures 11 and 12 respectively. Figures 13 and 14 present the results corresponding to an aspect ratio of 1.0. The maximum velocity is observed after distal anastomosis zone corresponding to 184 mm/s, 169 mm/s and 147 mm/s for the diameter aspect ratios equal to 0.7, 0.8 and 1.0 respectively. It can be seen that stagnation zones are larger for the bypass geometry corresponding to an aspect ratio of 0.7.
From the simulations it can be concluded that as the diameter aspect ratio increases the velocities decreases as well the pressures. A graft with a larger diameter produces smaller longitudinal velocity and smaller pressure in the host artery having positive effects for improving the hemodynamics of bypassing surgery.

4 CONCLUSIONS

Results from numerical modelling of blood flow through a bypass artery system using a developed finite element method are validated here by presenting velocity and pressure values obtained for different Reynolds numbers, using two viscosity laws and variable graft artery aspect ratio. By varying the aspect ratios from 1 to 0.7 this study indicates that pressures and velocities decrease by about 20%.

The outcomes will contribute to characterize the physiology of bypass grafts in circulatory system, and detect stagnation flow in recirculation zones. The objective of this preliminary study is also to test the capabilities of the developed mathematical model in order to build robust software for the optimization of bypass grafts shape aiming small gradient hemodynamic flow and minimizing recirculation zones.

Further studies could build more complicated computational models as considering unsteady flow and vessels deformation leading to even more realistic values for flow velocity and pressure profiles.

5 ACKNOWLEDGEMENTS

The authors thank the financial support by FCT, Portugal, project PTDC/SAU-BEB/102547/2008.

6 REFERENCES


