ENCIT-2018-0424
A STUDY ON THE INFLUENCE OF RHEOLOGY MODEL IN A CONSTRICTED CHANNEL FLOW

Gustavo P. V. Almeida

Instituto Tecnológico de Aeronáutica, DCTA/ITA, 12228-900, São José dos Campos, SP
gupalmeida@gmail.com

João Luiz F. Azevedo

Instituto de Aeronáutica e Espaço, DCTA/IAE/ALA, 12228-904, São José dos Campos, SP
joaoluiz.azevedo@gmail.com

Abstract. *The application of CFD to understand mechanical flow properties in complex flows plays an important role in hemodynamics. The hypothesis of using a Newtonian fluid in hemodynamics simulations is sometimes made in order to reduce modeling efforts. However, the blood is fundamentally a suspension of red blood cells in plasma, whose viscosity is mainly dependent of their volume fraction and how much these blood cells are deformed as a function of the flow properties. Hence, blood-like fluid structure may lead to a non-Newtonian behavior and neglecting these effects may jeopardize the analysis. The present work presents a study on the effects of the Herschel-Bulkley rheology model on the mechanical flow properties for a fluid flowing through two different constricted channel configurations using OpenFOAM. A sensitivity analysis revealed high impact of the rheology model μ_0 parameter as well as of changes in the geometry of the channel.*

Keywords: Rheology, Non-Newtonian Fluid, CFD, OpenFOAM, Constricted Channel

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is a field of science in which the governing equations of fluid motion are solved using numerical schemes. The CFD techniques consist fundamentally of applying these numerical schemes to study complex fluid flows. Complexities observed in fluid flows such as turbulence, non-linearity in fluid viscosity (rheology model), transient and non-stationary effects motivate the application of CFD in many fields such as in hemodynamics (Rosenfeld, 1993; Paul and Molla, 2012), oil and gas, aerodynamics, etc. The main idea behind numerical simulations is to visualize and understand these complex phenomena in flows where the application of analytical and experimental techniques are either not possible or prohibitively expensive.

The application of numerical methods for simulating the blood flow throughout the blood vessels is commonly known as Computational Hemodynamics (CHD) (Tu *et al.*, 2015). These techniques have been applied in the last decades in order to provide a better understanding of the blood flow properties. Moreover, the increasing interest in applying CHD for predicting possible cardiovascular diseases have been leading the scientific community to deeply study the subject. Therefore, there is a need for high-fidelity models in many different scales (Cebral *et al.*, 2007) and the rheology model, *i.e.*, the non-Newtonian behavior of the blood, is one of them.

The studies on the influence of rheology models over mechanical flow properties such as Wall Shear Stress (WSS), velocity profiles and pressure gradients play an important role in hemodynamics. The main idea relies on using CFD results as quantitative means combined with the qualitative current state-of-the-art medical features in the process of medical diagnostics (Wong and Poon, 2011). Many studies have been performed in an attempt of establishing a correlation between the blood flow properties and the development of the most critical cardiovascular diseases such as stenosis and aneurysms (Paul and Molla, 2012; Piskin and Celebi, 2013; Pinto *et al.*, 2013; Mamun *et al.*, 2015). Some researchers believe these cardiovascular diseases may be evaluated through the use of CFD prior to any surgical intervention.

The hypothesis of assuming the blood as a Newtonian fluid in hemodynamics simulations is sometimes made in order to simplify the models. Such an assumption is supported by the observations of only small variations on the blood throughout a given study, as stated by Tu *et al.* (2015). However, the blood is fundamentally a suspension of red blood cells in plasma, whose viscosity is mainly dependent of their volume fraction and how much these blood cells are deformed in function of the flow properties. In other words, the viscosity of the blood may change as a function of the flow properties such as pressure and strain rate and, therefore, neglecting these effects may jeopardize the analysis. Hence, the rheology model adopted may, in fact, impact the final results in hemodynamic simulations and the non-Newtonian assumption

should be made in order to obtain the desired high-fidelity model.

The main goal of this work is to present a study on the effects of the rheology model on the mechanical properties of the flow in a constricted channel using OpenFOAM. Newtonian fluid and blood-like non-Newtonian fluid models have been used for the studies in the present work. Mesh independence study is also presented hereafter.

2. METHODOLOGY

The authors have chosen to use the OpenFOAM (Weller *et al.*, 1998; OpenCFD, 2016) open source libraries for the numerical simulations of the flows of interest for their wide applicability for incompressible flows. The geometry is constructed using a simple constricted channel shape mainly because of its highly applicability in hemodynamic studies for modeling stenotic arteries. The authors describe here the theoretical and numerical formulations which are embedded in the tool used, with emphasis on the issues associated to the rheology model.

2.1 Theoretical Model

The fluid motion is described by the incompressible Navier-Stokes equations. These equations can be written as

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

and

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \, , \tag{2}$$

where u_j and u_i are the jth and i^{th} components of the velocity vector, respectively; ρ is the fluid density and τ_{ij} is the ij^{th} component of the viscous stress tensor. The viscous stress tensor for a Newtonian fluid is given by

$$\tau_{ij} = 2\mu e_{ij} \, , \tag{3}$$

where e_{ij} is the ij^{th} component of the strain tensor and μ is the fluid viscosity. The strain tensor, described in Eq. (3), may be written as

$$e_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \, . \tag{4}$$

Once the fluid model is assumed to be non-Newtonian the rheology model adopted for the present studies is one commonly used to describe a blood-like fluid viscosity, namely the Herscel-Bulkley model which combines both yield stress and shear-thinning characteristics (Valencia *et al.*, 2007; Kim, 2002). The Herschel-Bulkley model hypothesizes that the fluid viscosity changes as a function of the strain rate $\dot{\gamma}$, which by definition may be written as

$$\dot{\gamma} = \sqrt{2e_{ij}e_{ij}} \, . \tag{5}$$

The relation between the fluid viscosity and the strain rate is given by

$$\mu = k\dot{\gamma}^{n-1} + \frac{\tau_0}{\dot{\gamma}}, \tag{6}$$

where k is a constant of proportionality also called consistency index, n is the dimensionless flow index, and τ_0 is the yield stress. The model establishes a minimum τ_0 bias in the shear stress which the fluid shall exert in order to behave like a fluid. Otherwise, it behaves like a solid. The flow index indicates whether the fluid behaves like a shear-thinning or shear-thickening fluid. Blood-like fluids are treated as shear-thinning ($n < 1$) - also known as pseudo-plastic fluids, *i.e.*, the fluid viscosity decreases as the shear strain rate increases. On the other hand, the shear-thickening fluids ($n > 1$) - also known as dilatant fluids, have their viscosity increasing as the shear strain rate increases. Values of k and n for blood modeling are easily found in the literature (Tu *et al.*, 2015). Equation (6) clearly shows that for $n = 1$ and $\tau_0 = 0$ the model reconstitutes the original Newtonian formulation.

2.2 Numerical Formulation

The OpenFOAM is a set of C++ libraries encapsulated which basically perform vectorial and tensorial calculus. These libraries are applied to solve problems of fluid dynamics using the so-called Finite Volume Method (FVM) (Moukalled *et al.*, 2016) for discretization of the flow governing equations. There are several solvers available in the OpenFOAM library. However, the authors have chosen the *pimpleFoam* solver due to its wide variety of applicability (Almeida and Azevedo, 2017).

The spatial terms of the equations of interest are discretized and solved using a simple Gauss linear scheme. In other words, the integration of properties along the cell boundaries is made using the Gauss quadrature method with only one integration point, leading to a second order accurate scheme for the spatial terms. The numerical schemes available in the OpenFOAM libraries for temporal discretization are all implicit methods. The authors have chosen a simple first order Euler scheme for temporal discretization. The PIMPLE (PISO + SIMPLE) algorithm is used for solving the pressure-momentum coupling. The boundary conditions are defined as a fixed value of the mean flow velocity at the inlet and fixed value of pressure at the outlet for the steady case. The outflow velocity and inflow pressure are, then, calculated so that the flux at each boundary is respected.

The implementation of the Herschel-Bulkley fluid model in OpenFOAM follows a simple approach. As described in OpenCFD (2016), the model basically combines the effects of Bingham plastic fluids with the power-law behavior. The fluid will behave as a very viscous fluid with viscosity μ_0. The fluid viscosity changes in the presence of a strain-rate threshold in which the stress τ_0 is overcome. Hence, the fluid viscosity is obtained according to

$$\mu = min(\mu_0, k\dot{\gamma}^{n-1} + \frac{\tau_0}{\dot{\gamma}}) \tag{7}$$

The model implementation as per Eq. (7) avoids problems of singularities by choosing between the two given options.

2.3 Geometry Construction

The geometry considered for the calculations is a 20 mm diameter circular channel whose dimensions are referenced in Fig. 1. The preliminary solution is computed for two different constrictions: **a.** $d/D = 0.5$ and **b.** $d/D = 0.35$. The length L_1 is given so that the flow is fully developed upstream of the constriction and the length L_2 is defined so that the perturbations downstream of the constriction may vanish prior to leaving the domain. The relation for fully developed flow found for laminar incompressible flows is given by

$$\frac{L}{D} \approx 0.05 Re \tag{8}$$

where L is the length needed for the fully developed flow, D is the diameter and Re is the Reynolds number.
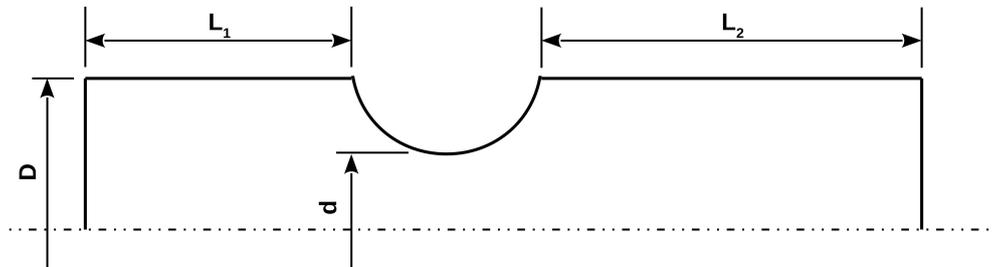

Figure 1. Dimensions of reference for geometry construction

2.4 Computational Grid

The computational grid has been generated using a commercial meshing tool. Figure 2 displays a typical mesh topology for these applications and also the one adopted for the simulations herein. The mesh is comprised mainly of unstructured hexahedral cells over the domain. The multiblock method was adopted as strategy for generating the computational meshes in the present study.

3. RESULTS

The studies of mesh independence were conducted using the Newtonian fluid model subjected to a steady flow condition with Reynolds number $Re = 200$. Five computational grids used. The number of mesh cells are $29,700$, $53,500$, $65,000$, $75,000$ and $137,500$, respectively. The results are presented in the Fig. 3. The figure above reveals convergence in the maximum flow velocity magnitude for mesh with number of cells higher than 75,000. The variation between the results presented for the two last mesh options is very low. Therefore, the results presented in this section are obtained for a computational mesh with $75,000$ cells, provided that the computational cost for the simulations is lower compared to the latter mesh option with $137,500$ cells.
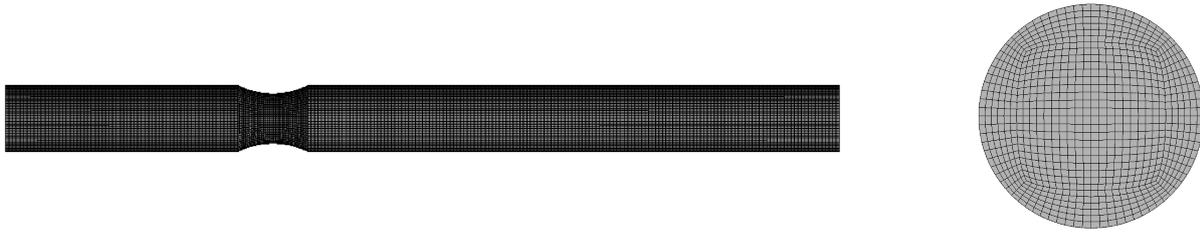
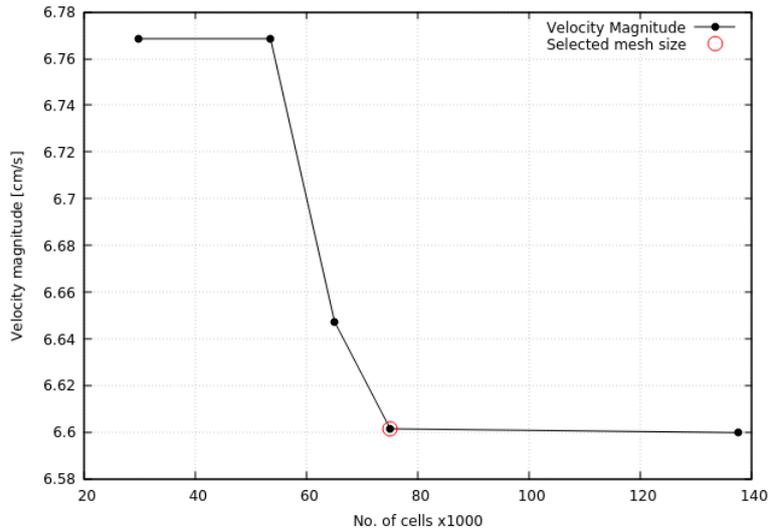Figure 2. Typical mesh topology for the flow of interest

Figure 3. Mesh refinement effects on the maximum flow velocity magnitude

All the simulations were conducted on a station with an Intel Core i5 processor, 8 Gb RAM memory and running on a 64-bit Ubuntu Linux operational system. The cases studied had five different configurations including changes in geometry and rheology model parameters. All the cases configurations are summarized in Tab. 1.

Table 1. Summary of cases setup

Case No.	Rheology Model	d/D	n	$\mu_0 \ [kgm^{-1}s^{-1}]$	$k \ [kgs^{n-2}m^{-1}]$
1	Newtonian	0.5	-	-	-
2	Herschel-Bulkley	0.5	0.7	3.465e-03	0.00622
3	Herschel-Bulkley	0.5	0.7	3.465e-02	0.00622
4	Newtonian	0.35	-	-	-
5	Herschel-Bulkley	0.35	0.7	3.465e-03	0.00622

A probe was placed upstream of the constriction in order to verify the effects of the rheology model prior to the disturbances caused by the constriction. The results show very small variations between cases 1 and 2 as per Fig. 4. The velocity contours indicate a behavior quite similar between the two models. However, sudden changes in the flow topology is observed in case 3. The modification imposed to μ_0 provided a more viscous behavior. Consequently, the region downstream of the constriction is severely affected. The length of the higher speed region is significantly reduced. One should observe that the color map used in all visualizations in Fig. 4 is the same.

The effects of the modifications in geometry are also presented. Cases 4 and 5 presented no variations in the rheology model parameters. The flow topology, however, is significantly impacted by the smaller relation of d/D. Velocity contours for these cases are depicted in Fig. 5.

The streamlines shown in Fig. 6 also provide a good visualization of impacts of the rheology models. The flow separation and recirculation bubbles can be observed. In general, the recirculation bubble zone extends for a much larger region in the case of the Newtonian fluid model.

The region of detachment and reattachment of the flow downstream of the constriction is similar in cases 1 and 2. Case 3 presents a very small recirculation region compared to the previous 2 cases. Cases 4 and 5 are also very different from each other in terms of streamlines visualization. The flow reattachment in case 4 occurs in a distance further than
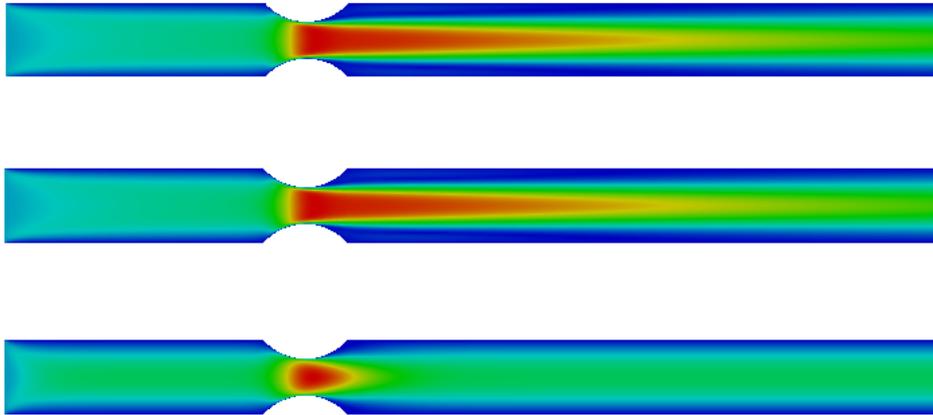
Figure 4. Velocity contours for cases 1, 2 and 3.
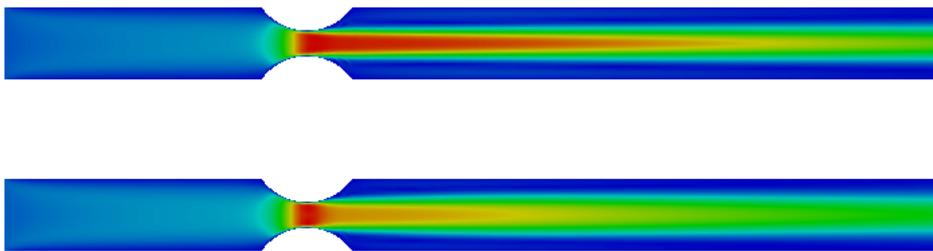
that presented in case 5.

Figure 5. Velocity contours for cases 4 and 5.

Table 2. Variation in maximum velocity magnitude for both rheology models.

Case No.	1	2	3	4	5
Maximum Velocity [cm/s]	16.43	16.43	17.02	30.82	27.87

Values of maximum flow velocity are taken from the constriction region where the flow reaches its maximum velocity. The constriction region is where the flow faces higher velocity gradients. Consequently, sudden variations in the shear strain-rates occur ending by changing the fluid viscosity. Evidence is noticed in the values presented for each of the cases in Tab. 2.

4. CONCLUSIONS

The present work provided a sensitivity analysis of the flow parameters in constricted channel flows when assuming a different rheology model. In general terms, the more realistic, non-Newtonian constitutive fluid model significantly affects the flow topology either upstream and downstream of the constriction, as opposed to a simpler Newtonian fluid assumption.

The non-Newtonian model presented great sensitivity to the variations in the μ_0 parameter. The fluid topology tends to become the same as that of the Newtonian model as μ_0 tends to small values. Variations in geometry (constriction) are also significant. Regions of the flow subjected to higher shear strain rates present high variations in the fluid viscosity. Consequently, the smaller relation of d/D presented the most affected flow topology given the rheology parameters and geometry simulated in cases 4 and 5.

In fact, the rheology model plays an important role in numerical simulations. The assumption of non-Newtonian fluid model considerably affects the final results, specially for geometries presenting abrupt variations such as presented in the human blood vessel anatomy. Moreover, the choice of fluid parameters (μ_0, k, n) shall be made carefully for obtaining more realistic results.

Figure 6. Stream lines for all the cases.

5. ACKNOWLEDGEMENTS

6. REFERENCES

Almeida, G.P.V. and Azevedo, J.L.F., 2017. "Simulation of a 2-D pulsatile flow in a bifurcated geometry using open source software". In *24th ABCM International Congress of Mechanical Engineering*. Curitiba, PR.

Cebral, J.R., Pergolizzi Jr., R. S. and Putman, C.M., 2007. "Computational fluid dynamics modeling of intracranial aneurysms: Qualitative comparison with cerebral angiography". *Academic Radiology*, Vol. 14, pp. 804 – 813.

Kim, S., 2002. *A Study of Non-Newtonian Viscosity and Yield Stress of Blood in a Scanning Capillary-Tube Rheometer*. Master's thesis, Drexel University, Philadelphia, PA.

Mamun, K., Akhter, M.N., Mollah, M.S.H., Sheikh, M.A.N. and Ali, M., 2015. "Characteristics of pulsatile blood flow through 3-D geometry of arterial stenosis". *Procedia Engineering*, Vol. 105, No. Supplement C, pp. 877 – 884. The 6th BSME International Conference on Thermal Engineering.

Moukalled, F., Mangani, L. and Darwish, M., 2016. *The Finite Volume Method in Computational Fluid Dynamics - An Advanced Introduction with OpenFOAM® and Matlab®*. Springer International Publishing Switzerland, Switzerland.

OpenCFD, 2016. *OpenFOAM User Guide Manual*. OpenCFD Limited.

Paul, M. and Molla, M., 2012. "Investigation of physiological pulsatile flow in a model arterial stenosis using large-eddy and direct numerical simulations". *Applied Mathematical Modelling*, Vol. 36, No. 9, pp. 4393–4413.

Pinto, S.I.S., Doutel, E., Campos, J.B.L.M. and Miranda, J.M., 2013. "Blood analog fluid flow in vessels with stenosis: Development of an OpenFOAM code to simulate pulsatile flow and elasticity of the fluid". *APCBEE Procedia*, Vol. 7, pp. 73–79.

Piskin, S. and Celebi, M.S., 2013. "Analysis of the effects of different pulsatile inlet profiles on the hemodynamical properties of blood flow in patient specific carotid artery with stenosis". *Computers in Biology and Medicine*, Vol. 43, pp. 717–728.

Rosenfeld, M., 1993. "Validation of numerical simulation of incompressible pulsatile flow in a constricted channel". *Computers & Fluids*, Vol. 22, No. 2, pp. 139–156.

Tu, J., Inthavong, K. and Wong, K.K.L., 2015. *Computational Hemodynamics - Theory, Modelling and Applications*. Springer Netherlands, Dordrecht, Netherlands.

Valencia, A., Botto, S., Sordo, J., Galvez, M. and Badilla, L., 2007. "Comparison of haemodynamics in cerebral aneurysms of different sizes located in the ophthalmic artery". *International Journal for Numerical Methods in Fluids*, Vol. 53, pp. 793–809.

Weller, H.G., Tabor, G., Jasak, H. and Fureby, C., 1998. "A tensorial approach to computational continuum mechanics using object-oriented techniques". *Computers in Physics*, Vol. 12, pp. 620–631.

Wong, G.K. and Poon, W., 2011. "Current status of computational fluid dynamics for cerebral aneurysms: The clinician's perspective". *Journal of Clinical Neuroscience*, Vol. 18, pp. 1285 – 1288.

7. RESPONSIBILITY NOTICE

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