The Application of Computational Fluid Dynamics (CFD) Method and Several Rheological Models of Blood Flow: A Review

Esmaeel FATAHIAN¹,*, Naser KORDANI², Hossein FATAHIAN¹

¹Young Researchers and Elite Club, Nour Branch, Islamic Azad University, Nour, Iran
²Department of Mechanical Engineering, University of Mazandaran, Mazandaran, Iran

Abstract
Computational fluid dynamics (CFD) method can be applied for gaining insights to the most fluid processes and related phenomena. Applying CFD method in the investigation of physiological flows especially blood is one of the interesting topics for many researchers. Because of its significant effect on various human cardiovascular diseases and arterial diseases, extended knowledge of blood flow in physiological conditions is required. This review provided an overview of recent studies on the application of CFD method of blood flow inside the corkscrew artery, arterial stenoses, human patient-specific left ventricle and arteries affected by multiple aneurysms. Also, several rheological models for describing the blood rheology were discussed. Based on this review, it was concluded that the application of CFD method can help the medical practitioners in the patients’ treatment decision in the investigation of blood flow.

1. INTRODUCTION

A fluid with properties which are different in any way from those of Newtonian fluids is a non-Newtonian fluid. Many molten polymers and salt solutions are non-Newtonian fluids including substances such as starch suspensions, (STF)/fabric composites, paint, ketchup, custard, maizena, toothpaste, blood and shampoo which are found many commonly [1,2]. Blood is a non-Newtonian fluid since its fluid properties are not described with a constant value of viscosity and the relationship between strain rate and shear stress is non-linear. Since the blood apparent viscosity reduces with increased stress, it is considered to be a shear thinning liquid. This caused in free flow at higher rates and low flow at slow rates of deformation [3]. The blood including semisolid components and some particles for example, the proteins, cells, ions, and lipoproteins [4]. Furthermore, several experimental studies concluded that the blood viscosity has in some range of the shear rate, the characteristics of thixotropic, shear thinning, viscoelasticity, and yield stress [5-7] and also the blood viscosity depends on the flow shear rate [8]. Blood is a suspension of white blood cells, red blood cells, nutrients, etc. in tubes with large internal diameters and in aqueous solution. The change in viscosity can be neglected when compared to the size of the red blood cells [9]. However, changes in apparent viscosity are evident in tubes with an internal radius of less than 0.5 mm [9]. In the aorta, the artery travelled by the blood is large and has high enough shear rates that it has been generally accepted for modeling blood flow as Newtonian [9,10]. The majority of studies conducted using blood as a non-Newtonian fluid take place in either bifurcated arteries or stenosed. This is intuitive to put the non-Newtonian properties of blood in both of these cases as the size of the artery which is remarkably smaller than for the aorta and the results vary due to the diameter change [11-14]. Many researchers have attracted the investigation of the blood flow over the past years. Because of its significant effect on various human cardiovascular diseases and arterial diseases, detailed knowledge of blood flow in physiological conditions is required. Nowadays, Applying CFD method in the investigation of physiological flows especially blood is an interesting topic for researchers [15-18]. In the analyzing of the blood flow mechanics in arteries indicates an important topic of vascular research and plays a significant role in the individual’s health. CFD models can be validated by measurement
techniques which are available [19-21] can be useful in the prediction of future disease progression [22]. Also, in order to describe the rheology of blood there are several non-Newtonian rheological models include Carreau-Yasuda [23,24], Casson [25], power law [26], Cross [27], Herschel-Bulkley [28], Oldroyd-B [29], Quemada [30], Yeleswarapu [31], Bingham [32], Eyring-Powell [33], and Ree-Eyring [34]. Besides, blood is modeled as a Newtonian fluid [25,35] and this approximation is good for large vessels with high shear rates in many circumstances. In the present study, the application of CFD method in different physiological conditions of blood flow and also several rheological models of blood flow are discussed.

2. THE USE OF CFD METHOD IN BLOOD FLOW

The prediction of fluid flow fields can be obtained by computational fluid dynamics based on numerical methods relating to continuity, momentum and energy equations [36]. Applying CFD method in the investigation of physiological flows especially blood is one of the interesting topics for many researchers. The assumption of non-Newtonian behavior of blood flow in CFD method has accepted for small vessels while Newtonian flow assumption is acceptable for large-sized arteries with high shear rates [37]. The validation of Newtonian assumption is still not completely clear for medium-sized vessels, especially in the stenotic case. Currently, for representing the viscous properties of blood, there is no universal agreement for the right model [38]. Consequently, modelling of blood non-Newtonian behavior is dramatically conducted and several non-Newtonian models are applied for studying their effects on blood flow characteristics including wall shear stresses, flow field, secondary flow patterns [39]. Table 1 illustrates the main researches using CFD method in blood flow.

<table>
<thead>
<tr>
<th>Authors</th>
<th>Methods</th>
<th>Modeling approaches</th>
<th>Operating/Boundary conditions</th>
<th>Findings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Doost et al. [8]</td>
<td>CFD</td>
<td>Carreau-Yasuda model, Casson model, K-L, Cross and Generalized power-law (GP)</td>
<td>Quasi-steady condition, the inlet and outlet boundaries was set pressure outlet and wall boundary condition</td>
<td>The local apparent viscosity had a large amount in the apex and the middle of the left ventricle (LV) in different non-Newtonian models, which caused to the low shear rate in these locations</td>
</tr>
<tr>
<td>Sharifi and Moghadam [40]</td>
<td>CFD</td>
<td>Carreau-Yasuda model</td>
<td>Incompressible fluid, laminar flow, velocity inlet boundary condition was used for the inlet of the artery and No-slip boundary condition for the vessel walls</td>
<td>The analyzing of blood flow in Buerger’s disease is significant because it helps better understanding of the possible prevention of disease and its progression</td>
</tr>
<tr>
<td>Manimaran [41]</td>
<td>CFD</td>
<td>Non-newtonian fluid model</td>
<td>Incompressible fluid, steady flow, No-slip boundary conditions on the stenosis wall</td>
<td>The pressure drop across a stenosed artery at low Reynolds numbers was unaffected by surface irregularities</td>
</tr>
<tr>
<td>Adib et al. [42]</td>
<td>CFD</td>
<td>Non-newtonian fluid model</td>
<td>Incompressible fluid, steady flow condition</td>
<td>They predicted the behavior of the blood flow velocity pattern under steady flow condition which can help the medical practitioners in their decision on the patients’ treatments</td>
</tr>
</tbody>
</table>
2.1. CFD study of blood flow inside the corkscrew artery

Buerger’s disease happens mostly in medium and small vessels due to the consume of Tobacco. One of the specifications of the Buerger’s disease is the existence of corkscrew collateral [40]. Fujii et al. [43] concluded that there is a relationship between the type of the corkscrew collateral arteries and clinical intensity of the Buerger’s disease and they classified the corkscrew collateral arteries according to the artery amplitudes. The CFD simulation of blood flow inside the corkscrew artery of the Buerger’s disease was conducted by Sharifi and Moghadam [40]. The blood characteristics which were applied in the CFD simulation were the same as the blood characteristics of a Buerger’s disease for a heavy smoker patient. In their study, the flow rate of blood was obtained from the experimental results of Van de Vosse et al. [44]. Figure 1 illustrates that the axial velocity profiles are in excellent agreement with the experimental results of Van de Vosse et al. [44] in Reynolds number of 300 at different cross-sections. Also, solid lines represent the numerical results of Sharifi and Moghadam [40] and spots show the experimental data of Van de Vosse et al. [44].

Their results indicated that the corkscrew artery geometry affects the blood flow patterns because it is very complex and cause to accumulate the flowing particles in blood like nicotine [40]. Moreover, many studies investigated the blood flow patterns impacts on stenting arteries and concluded that the hemodynamic can vary using different stent geometries [45]. As a result, the analyzing of blood flow in Buerger’s disease is significant because it helps better understanding of the possible prevention of disease and its progression.

2.2. CFD Study of blood flow in arterial stenosis

Previously, the flow situation in stenosis models had studied for different constrictions of the cross-sectional area and different Reynolds numbers. To solve numerically, the steady state condition sufficiently describes the behavior of pressure and flow around constrictions of blood vessels within the
arterial pipes [46]. Experimental and numerical studies of blood flow within arteries related to the formation of lesions and arterial narrowing have been done by many researchers [47,48]. Most of them, especially numerical modelling was concentrated on the carotid artery bifurcation [49,50]. Both in terms of simulation, this concentration was changed to the right coronary artery [51,52]. Manimaran [41] used CFD simulations in arterial stenoses with 48% areal occlusion and Non-Newtonian assumption was selected for the blood flow. In his study, three cases were considered for different Reynolds numbers. Figure 2 shows the pressure drop for various Reynolds number and viscosity models [41]. The Carreau model predicted the flow compared to the experiment results of Back et al. [46].

![Non-dimensional pressure based on Reynolds number](image)

**Figure 2.** Non-dimensional pressure based on Reynolds number [41]

Also, the flow resistance with the presence of surface irregularities was conducted. It was found that the pressure drop across a stenosed artery at low Reynolds numbers was unaffected by surface irregularities [41].

### 2.3. CFD study of blood flow in human patient-specific left ventricle

Although many researchers in their numerical study of the patient-specific left ventricle (LV) [53-55] ignored the impact of the blood non-Newtonian characteristic but the blood non-Newtonian characteristic might affect the flow properties of the patient-specific (LV) and this fact only conducted in simulations of a few researchers. For example, Schenkel et al. [56] used the Carreau-Yasuda model and Krittian et al. [57] applied the Carreau model for simulating the patient-specific LV. Doost et al. [8] simulated the imaged-based CFD of a patient-specific (LV) and to reconstruct the left ventricle (LV) geometry, the healthy human MRI images were applied. Figure 3 shows the schematic of the left ventricle (LV) and different boundary conditions. Both mitral and aortic valves were simulated as the orifices for capturing the movement of the valve leaflets because of the low spatiotemporal resolution of the MRI images [8].
They found that the local apparent viscosity had a large amount in the apex and the middle of the left ventricle (LV) in different non-Newtonian models, which caused to the low shear rate in these locations [8]. In the study of Adib et al. [42], a three-dimensional simulation was conducted to study the flow velocity pattern of heart regarding the left ventricle under steady flow condition. They investigated the velocity pattern of blood flow behavior under the steady condition that can help medical practitioners in patients’ treatments decision [42]. Bavo et al. [58] used a patient-specific CFD model of the mitral valve and left ventricle (LV) to verify the abnormal flow patterns for impaired hearts. They applied moving boundaries for three cardiac pathologies according to the transesophageal ultrasound images. Their CFD Results showed that the intraventricular flow dynamics were consistent with diagnostic patient records. The combination of clinical images and CFD method caused to investigate deeply the intraventricular hemodynamics in pathophysiology [58]. A novel CFD study of the left ventricle (LV) was done in the study of Larsson et al. [59]. In their CFD modeling, a deviation of below 12% was obtained and it was evaluated the simulation pathway sensitivity when input parameters varied including evaluating clinician or image acquisition. Also, the average errors between CFD simulation and clinically acquired in-vivo flow measurements indicated below 11% so it verified the accuracy of the simulation pathway. Generally, applying patient-specific models may work as a precious tool for three-dimensional blood flow motion in future clinical researches because these models offer the detailed investigation of isolated flow phenomena in direct relation to defined pathologies [59].

2.4. CFD study of blood flow in arteries affected by multiple aneurysms

Several authors accomplished the various aneurysm geometries of patients over the last decade [60,61]. Wille [62] numerically studied the blood flow for tracing streamlines in moderately dilated rigid blood vessels. Also, Perktold [63] analyzed the path lines of the flow particles and Kumar and Naidu [64] investigated single aneurysm for different dilations. Arterial diseases are not fully understood due to hemodynamic factors. Bai et al. [65] applied CFD method for numerical simulations of affected blood vessels both in steady and unsteady conditions in arteries affected by multiple saccular aneurysms. In blood vessels, finite volume method (FVM) was used for studying the hemodynamic factors including streamline patterns, pressure, velocity and shear stress of wall. CFD method is significant for detecting...
and quantifying of multiple aneurysms act as a tool for surgical interference [65]. In order to provide a relatively low-cost alternative for benching top modeling, computer simulation models are useful for growth, rupture and development of an aneurysm and yet they have been criticized due to the lack of experimental validation. Moreover, the combination of CFD method and three-dimensional x-ray angiography which obtained by the application of biomorphometric techniques and CFD method are current approaches [66,67]. Lott et al. [68] investigated the flow properties in saccular cerebral aneurysm by using FVM method as a 2D model. They found that in order to prepare objective data on biophysical parameters, computer simulation of aneurysm flow can obtain flow events in vitro aneurysms when carefully developed [68]. Impressive flow simulation of aneurysms depends on the size of the parent vessel and aneurysm, flow input and etc. The important characteristics of flow include neck and proximal dome configuration and independent of size [68].

3. DIFFERENT RHEOLOGICAL MODELS OF BLOOD FLOW

In describing the blood rheology, there are several non-Newtonian rheological models including Carreau-Yasuda [23,24], Casson [25], power law [26], Cross [27], Herschel-Bulkley [28], Oldroyd-B [29], Quemada [30], Yeleswarapu [31], Bingham [32], Eyring-Powell [33], and Ree-Eyring [34]. Also, other fluid models which are less known [69] have applied for describing the blood rheology. The non-Newtonian relation of power-law model is as follows [70,71]:

$$\tau = m\gamma^n$$  

(1)

The constants of power-law model are $n$ and $m$ which represent the fluid consistency and the degree of non-Newtonian behavior, respectively.

Also, the power-law fluid viscosity is as follows [72]:

$$\eta = m\gamma^{n-1}$$  

(2)

The non-Newtonian apparent viscosity is $\eta$. By increasing the shear rate and significantly decreasing apparent viscosity, a shear thinning fluid is generated for $n < 1$. Also, by increasing the shear rate and progressively increasing apparent viscosity, a shear-thickening fluid is generated for $n > 1$ [73]. Finally, we have a Newtonian fluid for $n = 1$. In Figure 4, the flow curves of these power-law models have shown. The power-law model cannot describe many non-Newtonian fluids viscosity in locations where the shear rate is very low and very high and this is an obvious disadvantage of power-law model.

Figure 4. Flow curves of power-law fluids: (a) shear-thinning fluid, (b) Newtonian fluid, (c) shear-thickening fluid [70,71]
In non-Newtonian modeling, Carreau-Yasuda and Casson are the most popular models of blood flow which are reported in many researches. Besides, blood is modeled as a Newtonian fluid \([25,35]\) and this approximation is good for large vessels with high shear rates in many circumstances under non-pathological conditions \([74]\). Analytical \([75]\), stochastic \([76,77]\), and numerical mesh methods; such as finite element \([78]\), finite difference \([79]\), finite volume \([80]\), and spectral collocation methods \([81]\) are diverse methods. In simulating and modeling non-Newtonian effects in blood rheology, these methods were used. The lattice Boltzmann method for simulation of the non-Newtonian shear-dependent viscosity models was used in the study of Boyd et al. \([82]\). For comparison of the two-dimensional non-Newtonian and Newtonian flows in both oscillatory and steady flows, the non-Newtonian blood viscosity models including Carreau-Yasuda and Casson were applied in curved and straight pipe geometries \([82]\). It was concluded that both the Carreau-Yasuda and Casson flows indicate remarkable differences in the steady flow situation compared to analogous Newtonian flows and these differences are significant for the investigation of atherosclerotic progression \([82]\). Sankar et al. \([83]\) considered the impacts of blood non-Newtonian behavior, stenosis, and pulsatility. They investigated the pulsatile blood flow in stenosed artery and assumed the blood as a Herschel-Bulkley fluid. For analyzing the flow, a perturbation method was applied and the plug core region was assumed to change with axial distance. Figure 5 illustrates that the plug core radius reduces for the values of \(n, A, and \delta_s\) as the radius \(R\) decreases \([83]\).

**Figure 5. Variation of plug core radius \([83]\)**

Furthermore, they concluded that the pressure gradient increased with the increment of yield stress in steady flow and for a given flow rate, the Herschel-Bulkley fluid had more pressure gradient compared to the power law fluid \([83]\). Also, Sankar et al. \([84]\) modeled blood as a two-fluid model including Newtonian and Herschel-Bulkley fluid. They studied the steady flow of blood through a catheterized artery. The catheter was inserted coaxially due to assuming the artery as a rigid wall and wall shear stress, flow rate, velocity, and frictional resistance parameters were analyzed \([84]\). Figure 6 to 9 indicate these parameters based on catheter radius ratio yield stress and peripheral layer thickness. As can be seen, by increasing the thickness of the peripheral layer, the frictional resistance and the wall shear stress decreased and flow rate and velocity increased. Moreover, when the catheter radius ratio or yield stress increased, the wall shear stress increased while the flow rate and velocity decreased \([84]\).
Figure 6. Velocity distribution [84]

Figure 7. Variation of wall shear stress [84]

Figure 8. Variation of flow rate [84]
They found that by increasing the yield stress, the width of the plug flow region increased. As a result, in order to investigate the blood flow through catheterized arteries, the study of Sankar et al. could be effective [84].

4. CONCLUSION

The application of CFD method in different physiological conditions of blood flow and also several rheological models of blood flow are discussed in this study. The assumption of non-Newtonian behavior of blood flow in CFD method has accepted for small vessels while Newtonian flow assumption is acceptable for large-sized arteries with high shear rates. Currently, for representing the viscous properties of blood there is no universal agreement for the right model. Consequently, modeling of blood non-Newtonian behavior is dramatically conducted and several non-Newtonian models are applied for studying their effects on blood flow characteristics. The analyzing of blood flow in Buerger’s disease is significant because it helps better understanding of the possible prevention of disease and its progression. Although many researchers in their numerical study of the patient-specific left ventricle (LV) ignored the impact of the blood non-Newtonian characteristic but the blood non-Newtonian characteristic might affect the flow properties of the patient-specific (LV) and this fact only conducted in simulations of a few researchers. Also, a three-dimensional simulation for studying the flow velocity pattern of heart regarding the left ventricle under steady condition can help the medical practitioners in patients’ treatments decision. Knowledge of pressure, flow partition and shear stress caused to have a better understanding of the relationship between the fluid dynamics within pulsatile blood flow and arterial diseases. Also, in order to describe the blood rheology, there are several non-Newtonian rheological models. Besides, blood is modeled as a Newtonian fluid and this approximation is good for large vessels with high shear rates in many circumstances. The power-law model cannot describe many non-Newtonian fluids viscosity in locations where the shear rate is very low and very high and this is an obvious disadvantage of power-law model. In non-Newtonian modeling, Carreau-Yasuda and Casson are the most popular models of blood flow which are reported in many researches. As a result, both the Carreau-Yasuda and Casson flows indicate remarkable differences in the steady flow situation compared to analogous Newtonian flows and these differences are significant for the investigation of atherosclerotic progression.

CONFLICTS OF INTEREST

No conflict of interest was declared by the authors.

REFERENCES


