3D Simulations of Blood Flow Dynamics in Compliant Vessels: Normal, Aneurysmal, and Stenotic Arteries

Yongsam Kim\(^1\),*, Yunyoung Park\(^1\) and Sookkyung Lim\(^2\)

\(^1\) Department of Mathematics, Chung-Ang University, Dongjakgu Heukseokdong, Seoul 156-756, Korea.
\(^2\) Department of Mathematical Sciences, University of Cincinnati, Cincinnati, Ohio, 45242, USA.

Received 17 February 2015; Accepted (in revised version) 30 December 2015

Abstract. Arterial diseases such as aneurysm and stenosis may result from the mechanical and/or morphological change of an arterial wall structure and correspondingly altered hemodynamics. The development of a 3D computational model of blood flow can be useful to study the hemodynamics in major blood vessels and may provide an insight into the noninvasive technique to detect arterial diseases in early stage. In this paper, we present a three-dimensional model of blood flow in the aorta, which is based on the immersed boundary method to describe the interaction of blood flow with the aortic wall. Our simulation results show that the hysteresis loop is evident in the pressure-diameter relationship of the normal aorta when the arterial wall is considered to be viscoelastic. In addition, it is shown that flow patterns and pressure distributions are altered in response to the change of aortic morphology.

AMS subject classifications: 76D05, 66Z05, 92C35

Key words: Blood flow, immersed boundary method, compliant vessel, aortic aneurysm, aortic stenosis.

1 Introduction

The aorta is the largest artery that carries oxygen-rich blood from the left ventricle of the heart and runs down to the abdomen [39]. In healthy human individuals, the aorta normally takes the shape of a candy cane, and the primary constituents of the aortic wall are known as elastin, collagen, smooth muscle, and ground substance [9,55]. Note in general that collagen is stiffer than elastin. The close association of those components determines

*Corresponding author. Email addresses: kims@cau.ac.kr (Y. Kim), yunypark@cau.ac.kr (Y. Park), sookkyung.lim@uc.edu (S. Lim)

the mechanical properties of the aorta, in particular, its viscoelasticity that accounts for many mechanical features of the aorta. The aortic wall expands during systole (contraction of the heart muscle) and contracts during diastole (relaxation of the heart muscle). The motion of the aortic wall is closely related to the aortic pressure as a result of the non-linear elasticity of the arterial wall [1, 2, 4, 9, 10]. Variations in both the elastic properties of the artery and the blood pressure inside the artery are crucial factors in early stage of the development of potential cardiovascular diseases [10, 36, 37].

There have been various approaches to explore the interaction between the blood flow and the arterial wall, in which models with different spatial dimensions are used for such exploration. One-dimensional (1D) models have been proposed by averaging the fluid properties over the cross-section of the vessel [7, 8, 18, 45, 48]. Two-dimensional (2D) models have been used to address the interaction between the incompressible viscous fluid and the deformable viscoelastic wall, where the blood vessel is assumed to be axially symmetric [10, 11, 23]. Another approach is to use three-dimensional (3D) models for the blood inside the compliant wall where the displacement of the wall is described by an appropriate boundary condition [5, 12, 13, 31, 33, 40, 47, 53]. There are also multi-scale models that couple 3D models with 1D models associated with appropriate coupling conditions [17, 19, 38].

In this paper we present a mathematical model which can predict the flow velocity and pressure of blood in the aorta in relation to the mechanical and geometrical properties of the arterial wall. We have previously applied the immersed boundary (IB) method to a 2D model of blood flow in a compliant vessel [29] and have shown that the method is well suited for the interaction between the blood and a moving vessel wall. However, the results in [29] and generally in 2D simulations are inherently artificial. In fact, it is difficult in 2D simulations to evaluate the wall properties in a realistic way and to investigate the relation between the elastic properties of the vessel wall and the blood flow, which is the main purpose of this research. Here we extend and generalize the IB method used in [29] to a full three-dimensional model of the fluid-structure interaction. In the IB formulation, since the action of the deformable elastic vessel wall appears as a localized body force acting on the blood flow, the Navier-Stokes solver does not need to know anything about the complicated time-dependent geometry of the elastic vessel, and therefore we can avoid the difficulties caused by the fluid-structure interaction.

One of the main features of the IB method [42, 44] is that the method enables us to model the structure of the arterial wall by imitating closely its anatomy. In our model, the aortic wall is composed of several layers, each of which is then made up of elastic fibers. This fiber structure is observed in the actual arterial wall [1] and has been shown to be an optimal one for a blood vessel [4]. A source and a sink are used to prescribe the flow rate at the inlet and the blood pressure at the outlet, respectively, based on the experimental data measured in healthy human subjects [29, 39].

Hysteresis behavior, in general, is a property of physical systems, in which the system reacts to the applied force with a time delay. This hysteresis phenomenon is apparently observed in the relationship between the blood pressure and the vessel diameter [9]. The
potential physical parameters that might cause the hysteresis in the blood flow dynamics are fluid viscosity, vessel wall density, and damping coefficient of the vessel (viscosity of the arterial wall) [29]. It turns out that the damping coefficient is the only factor among these that generates the hysteresis loop in the viscoelastic arterial wall. This has been reported in literature [1,2,4,10,29], and here we present the relation between the damping coefficient and hysteresis behavior.

We also present some extensions of the 3D vessel model which describe the hemodynamics of aneurysmal or stenotic aorta. An aneurysm, which is a dilation of blood vessels, may occur in brain and coronary arteries, or at any part of the aorta; for example, the aortic root and descending aorta. In contrast, a stenosis is an abnormal narrowing in a blood vessel often caused by atherosclerosis [39]. We consider five different configurations as follows: a straight but tapered artery as a normal case, symmetrically and asymmetrically bulging arteries as aneurysmal cases, and symmetrically and asymmetrically constrictive arteries as stenotic cases. We shall compare the flow pattern and pressure difference inside the arterial wall for these different configurations.

2 Equations of motion

To begin with, we state a mathematical formulation for a coupled system of a three-dimensional viscous incompressible fluid interacting with an immersed elastic structure which represents a vessel wall in our case. In the IB framework, the fluid exists not only inside the vessel but also outside the vessel. Although the outside region of the vessel is filled with tissues or other organs, we assume that the same fluid is everywhere. While this assumption might be a limitation for some applications such as blood vessels in bone or in the lung, and blood vessels close to the skin, it is quite reasonable for the problem of a typical blood flow. This is because the tissue outside the typical blood vessel has about the same density as blood. However, the inertial and viscous loads provided by the external tissue may play an important role in the hemodynamics, and such loads can be implemented in the model as a pressure boundary condition [10, 18]. Alternatively, one can model the external material by using elastic fibers in the space external to the vessel, which is standard in the IB method.

The system of governing equations reads as follows:

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) + \nabla p = \mu \nabla^2 \mathbf{u} + \mathbf{f}, \quad (2.1) \]

\[ \nabla \cdot \mathbf{u} = \sum_{j=1}^{m} Q_j(t) \psi_j(\mathbf{x}), \quad (2.2) \]

\[ \mathbf{F} = -\frac{\partial E}{\partial \mathbf{X}} - c \frac{\partial \mathbf{X}}{\partial t}(r,s,t), \quad (2.3) \]
\[ f(x,t) = \int F(r,s,t)\delta(x - X(r,s,t))dqdrds, \quad (2.4) \]

\[ \frac{\partial X}{\partial t}(r,s,t) = u(X(r,s,t),t) = \int u(x,t)\delta(x - X(r,s,t))dx. \quad (2.5) \]

Eqs. (2.1) and (2.2) are the Navier-Stokes equations for a viscous incompressible fluid except that the continuity equation is modified to allow creation and/or destruction of the fluid. This modification is needed to drive the flow through the vessel and to regulate the transmural pressure difference along the vessel. The function \( \psi_j(x) \) is a spatial weight function for which the integral over the whole fluid domain is 1, and \( Q_j(t) \) is the volumic flow rate. A positive value of \( Q_j(t) \) represents a source and a negative value represents a sink. This source/sink methodology has previously been used to drive flow through a vessel, see [3, 46]. The constant parameters \( \rho \) and \( \mu \) are the fluid density and viscosity, respectively. The unknown functions in the fluid equations are the fluid velocity, \( u(x,t) \); the fluid pressure, \( p(x,t) \); and the force per unit volume applied by the immersed structure to the fluid, \( f(x,t) \), where \( x = (x,y,z) \) are fixed Cartesian coordinates, and \( t \) is the time.

Eq. (2.3) is the immersed structure equation which is written in Lagrangian form. The unknown \( X(r,s,t) \) completely describes the motion of the immersed structure, and also its spatial configuration at any given time \( t \). The vessel wall of our model is a network of elastic fibers, and thus the variable \( r \) of \( X(r,s,t) \) represents an index of each fiber and the variable \( s \) is a parameter that varies along each fiber, see Section 3 for a detailed description. In Eq. (2.3), \( F(r,s,t) \) is the force density applied by the immersed structure to the fluid, in the sense that \( F(r,s,t)dqdrds \) is the force applied to the fluid by the differential patch of immersed structure \( dqdrds \). The elastic contribution to this force density is given by the variational derivative \(-\delta E/\delta X\) of the elastic energy functional \( E[X(\cdot,\cdot,t)]\). This variational derivative is implicitly defined by

\[ dE(t) = \int \frac{\partial E}{\partial X}(r,s,t) \cdot dX(r,s,t)dqdrds, \quad (2.6) \]

where \( dX \) is a perturbation of the structure configuration and \( dE \) is the resulting perturbation in the elastic energy of the structure (to first order). The second term of the right side of Eq. (2.3) is the damping term which represents the viscoelastic property of the arterial wall [10, 18, 29].

Eqs. (2.4) and (2.5), which are called interaction equations, express the local character of the interaction via a three-dimensional Dirac delta function \( \delta(x) = \delta(x)\delta(y)\delta(z) \). Eq. (2.4) expresses the relation between the two corresponding force densities \( f(x,t)dx \) and \( F(r,s,t)dqdrds \) which can be seen by integrating each side of Eq. (2.4) over an arbitrary fluid region \( \Omega \). Eq. (2.5) is the equation of motion of the immersed elastic structure. It is the no-slip condition which says that the structure moves at the local fluid velocity. This is rewritten in terms of the Dirac delta function in the second form of Eq. (2.5).
3 Three-dimensional vessel model with source/sink

In this section we introduce a three-dimensional model of a blood vessel to study the flow dynamics. We present the initial setting of our model, the way of driving flow using the source/sink, and physical and computational parameters used in the numerical experiments.

Consider an incompressible viscous fluid in a rectangular box \([-3.2,3.2] \times [-3.2,3.2] \times [0,12.8] \text{ cm}^3\), in which a compliant vessel is immersed and interacts with the surrounding fluid. The vessel wall is comprised of five layers, each of which takes the shape of a tapered cylinder built by a collection of elastic fibers. The innermost layer has the radius of \(R_{up}\) at the upstream end, and the radius decreases linearly to be \(R_{dw}\) at the downstream end. The other layers are constructed in the same fashion with increasing radii so that the thickness of the vessel becomes \(D\). Fig. 1 shows the innermost and outermost layers of the vessel in the perspective view (left) and on its 2-D cross section (middle) in the \(xz\)-plane containing the center line of the 3D vessel.

Each layer of the vessel wall is made up of three different types of elastic fibers: longitudinal, helical, and circular fibers, see the left panel of Fig. 1. The fibers and thus the vessel wall are elastic materials from which the force is generated by Eqs. (2.3) and (2.6),
in which the elastic energy $E(X)$ of the fibers is of the form
\[
E(X) = \frac{1}{2} s \int \left( \left| \frac{\partial X}{\partial s}(r,s,t) \right| - 1 \right)^2 dr ds.
\] (3.1)

Here the variable $s$ in $X(r,s,t)$ is the parameter for each fiber, $r$ represents the index of different fibers, and $t$ is the time. Eq. (3.1) says that the elastic energy is generated when the fibers on each layer are stretched or compressed from their rest state and that there is no energy generated from inter-layers and inter-fibers.

The vessel wall is slightly extended and annexed to rigid caps at the top and bottom of the vessel, see the left and middle panels of Fig. 1. Here the rigid caps are tethered in space, and hence we can model them by using the idea of “target boundary,” as follows [29, 42]. Define $Y(r,s,t = 0)$ as the initial configuration of the rigid caps and their target boundary at the same time. Then the time-dependent immersed boundary $Y(r,s,t)$ should be kept as close as possible to the target boundary $Y(r,s,t = 0)$. This can be done by generating a strong restoring force $c_k (Y(r,s,0) - Y(r,s,t))$ and applying it to the fluid as the body force. The constant $c_k$ can be chosen to make the deviation $|Y(r,s,0) - Y(r,s,t)|$ to be as small as we like.

Inside the rigid caps located at the top and bottom are the regions $\Omega_{\text{up}=1}$ and $\Omega_{\text{dw}=2}$, respectively, on which the source/sink is used to derive the blood flow in the vessel, see the middle panel of Fig. 1. We also prescribe an external source/sink on the region $\Omega_{\text{ext}=0}$ outside the vessel in order to preserve the total fluid volume in the whole computational domain. The region $\Omega_{\text{ext}=0}$ is chosen to be four rectangular columns placed at four corners of the computational box. The spatial distribution of $\Omega_{\text{ext}}$’s can be identified by defining three weight functions $\psi_j(x)$, $j = 0,1,2$, which are bell-shaped with positive values over the corresponding region $\Omega_j$ and zero outside. By choosing $\psi_j(x)$ such that $\int \psi_j(x) dx = 1$, we ensure that $Q_j(t)$ can be interpreted as the total flux through the region $\Omega_j$. Since we use the periodic boundary condition, it is required that the total flow rate is zero, i.e., $\int \nabla \cdot \mathbf{u}(x,t) dx = 0$, which implies $\sum_{j=0}^{2} Q_j(t) = 0$ for all time. We meet this requirement by equating $Q_0 = -(Q_1 + Q_2)$.

In the numerical experiments presented in this work, the flux $Q_1(t)$ on the region $\Omega_{\text{up}=1}$ is determined based on the desired longitudinal velocity $w_1(t)$ of the vessel which was obtained from Magnetic Resonance (MR) imaging data measured in the thoracic aorta of a healthy subject over one cardiac cycle [29], see the right-top panel of Fig. 1. Assuming that the flow in the upstream region is a Poiseuille flow in a rigid cylinder with the radius $R_{\text{up}}$, we obtain the downward velocity as $w_1(t) (R_{\text{up}}^2 - r^2) / R_{\text{up}}^2$, where $r$ is the radial distance from the center of the cylinder, and we can compute the downward flux in the upstream region of the vessel to be $\pi R_{\text{up}}^2 w_1(t)/2$. Then we choose the flux as twice the downward flux, i.e., $Q_1 = \pi R_{\text{up}}^2 w_1(t)$.

In the region $\Omega_{\text{dw}=2}$, the flux $Q_2$ is determined so that the pressure level in the region $\Omega_2$ is the same as the blood pressure level $\tilde{\rho}(t)$ of a real human heart which was taken from [39] as shown in the right-bottom panel of Fig. 1 over one cardiac cycle, 1 sec. Since
\( \psi_2(x) \) is zero outside the region \( \Omega_2 \) and \( \int \psi_2(x) \, dx = 1 \), we can compute the pressure in \( \Omega_2 \) by
\[
\bar{p}_{dw}(t) - \bar{p}_{ext}(t) = \int p(x,t) (\psi_2(x) - \psi_0(x)) \, dx.
\] (3.2)

By setting \( \bar{p}_{dw}(t) - \bar{p}_{ext}(t) \) to be the same as the desired blood pressure level \( \tilde{p}(t) \), we can uniquely determine the flow rate \( Q_2(t) \) in the region \( \Omega_2 \), see below in Section 4 for more details.

Table 1 shows physical parameters used in our simulations. The fluid density and viscosity are those of the typical blood in major arteries. The dimensions of the vessel such as arterial radius, length, and wall thickness are chosen in the range of those of the aorta measured in a healthy subject [29, 35, 39, 41]. Note that the arterial wall has larger radius at the upstream than at the downstream. The stiffness coefficient \( c_s \) is close to the value observed in literature [10, 18, 39]. The damping coefficient \( c_r \) which represents the viscoelastic property of the arterial wall is known to cause hysteresis behavior of the blood flow in compliant arteries [10]. Canic et al. [10] derived the viscous modulus \( c_r \) to be \( 1.6 \times 10^8 \) (dyne-s/cm^2) for the human femoral artery which is of the same order of magnitude as the measurements corresponding to the dog's aorta reported in [1]. We vary the damping coefficient \( c_r \) to see the dependence of the hysteresis on the viscoelastic material property.

### Table 1: Physical parameters.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Magnitude</th>
</tr>
</thead>
<tbody>
<tr>
<td>fluid density, ( \rho )</td>
<td>1.05 g/cm^3</td>
</tr>
<tr>
<td>viscosity, ( \mu )</td>
<td>0.04 g/(cm \cdot s)</td>
</tr>
<tr>
<td>artery length, ( L )</td>
<td>10.0 cm</td>
</tr>
<tr>
<td>artery thickness, ( D )</td>
<td>0.15 cm</td>
</tr>
<tr>
<td>upstream artery radius, ( R_{up} )</td>
<td>1.0 cm</td>
</tr>
<tr>
<td>downstream artery radius, ( R_{dw} )</td>
<td>0.825 cm</td>
</tr>
<tr>
<td>stiffness coefficient, ( c_s )</td>
<td>( 4 \times 10^6 ) dyne/cm^2</td>
</tr>
<tr>
<td>damping coefficient, ( c_r )</td>
<td>( 0 - 6 \times 10^4 ) (dyne-s/cm^2)</td>
</tr>
</tbody>
</table>

Once we construct the initial configuration of the vessel model and set the physical parameters, we solve Eqs. (2.1)-(2.5) by following the numerical procedure described in Section 4. The meshwidth of the computational domain, \( \Delta x = \Delta y = 6.4/64 \) cm, is uniform and fixed in time, and the time duration is \( \Delta t = 5 \times 10^{-5} \) s. The spring constant to enforce the rigid caps to be fixed in space is set equal to \( c_k = 5 \times 10^8 \) dyne/cm^2. This choice of \( c_k \) makes the maximum deviation of the immersed boundary \( Y(r,s,t) \) from the target boundary \( Y(r,s,0) \) to be less than \( \Delta x/5 \), i.e., \( \| (Y(r,s,0) - Y(r,s,t)) \|_\infty < \Delta x/5 \).
4 Numerical implementation

For the numerical implementation to solve Eqs. (2.1)-(2.5), we use a formally second-order IB method which is generalized to take into account the sink and source terms [27, 43, 46]. The formally 2nd order scheme was developed in [34, 43] and shown numerically to be 2nd order accurate for problems with sufficiently smooth solutions in [22, 28]. Even though our solutions are not smooth (the velocity has jumps in derivative across the immersed boundary), the use of the formally 2nd order method results in improved accuracy, i.e., the scheme generates a smaller numerical viscosity and thus thinner boundary layer than 1st order accurate schemes [34, 43].

Let a superscript \( n \) denote the time level; thus, \( X^n(q,r,s) \) is a shorthand for \( X(r,s,n\Delta t) \), where \( \Delta t \) is the duration of the time step, and similarly for all other variables. The formally second-order IB method follows a 2-stage Runge-Kutta framework: (i) a preliminary step is processed to obtain data at time level \( n+1/2 \) from data at time level \( n \) by a first-order accurate method, and (ii) a final substep starts again at time level \( n \) and proceeds to time level \( n+1 \) by a second-order accurate method. The step-by-step procedure of the numerical implementation can be summarized as follows:

(I) Update the position of a vessel to obtain \( X^{n+1/2} \) at time level \( n+1/2 \). This is done by the discretization of Eq. (2.5).

(II) Using the position \( X^{n+1/2} \) and velocity \( U^{n+1/2} \) of the vessel, calculate the Lagrangian force density \( F^{n+1/2} \), see Eq. (2.3). The vessel velocity \( U^{n+1/2} \) is computed by Eq. (2.5).

(III) Change this elastic force density defined on Lagrangian grid points into the fluid force density \( f^{n+1/2} \) at Eulerian spatial grid points which will be applied in the Navier-Stokes equations. This is done by a discretization of Eq. (2.4).

(IV) Given the Eulerian force density \( f^{n+1/2} \), we solve the discretized version of the fluid equations (2.1) and (2.2):

\[
\rho \left( \frac{u^{n+1/2} - u^n}{\Delta t/2} + W(u^n) \right) + Dp^{n+1/2} = \mu L(u^{n+1/2} + u^{n+1/2}) + f^{n+1/2}, \quad (4.1)
\]

\[
D \cdot u^{n+1/2} = \sum_{j=0}^{2} \psi_j(x) \quad (4.2)
\]

(V) Update the position of the vessel at time level \( n+1 \) in the same manner as done in (I).

(VI) Update the fluid velocity data by solving the fluid equations:

\[
\rho \left( \frac{u^{n+1} - u^n}{\Delta t} + W(u^{n+1/2}) \right) + Dp^{n+1} = \frac{1}{2} \mu L(u^{n+1} + u^n) + f^{n+1}, \quad (4.3)
\]

\[
D \cdot u^{n+1} = \sum_{j=0}^{2} \psi_j(x) \quad (4.4)
\]

In the discretized equations (4.1)-(4.4) above, \( D = (D_1^0, D_2^0, D_3^0) \) and \( D_0^1 \) is the standard central difference operator in the spatial direction denoted by \( i \), where \( i = 1,2,3 \), and \( L \) is
the standard 5-point discrete Laplacian. The convection terms $W(u^n)$ and $W(u^{n+\frac{1}{2}})$ are treated explicitly and approximated by the 2nd order ENO scheme [24, 49, 50].

Eqs. (4.1)-(4.2) and (4.3)-(4.4) are linear systems with constant coefficients for the unknowns $u^{n+\frac{1}{2}}, \rho^{n+\frac{1}{2}}$ and $u^{n+1}, p^{n+\frac{1}{2}}$, respectively. We solve these linear systems by taking its discrete Fourier transform (implemented by the FFT algorithm). In order to solve these systems, however, we need to know $Q_j^{n+\frac{1}{2}}$ and $Q_j^{n+1}$. As described in the previous section, while the flux $Q_1(t)$ in the upstream region is prescribed to be $\pi R_{up}^2 w_1(t)$, the flux $Q_2(t)$ is to be determined so that the blood pressure level of a real human heart is the same as the average pressure level in the downstream region $\Omega_2$. In order to solve Eqs. (4.1)-(4.2) with the condition of the blood pressure level, using the fact that Eqs. (4.1)-(4.2) are linear for the unknown variables $u^{n+\frac{1}{2}}$ and $\rho^{n+\frac{1}{2}}$, we first divide the system into the following three systems (A) - (C) (Eqs. (4.3)-(4.4) can be solved in the same fashion):

(A)

$$\begin{align*}
\rho \left( \frac{u_f^{n+\frac{1}{2}} - u^n}{\Delta t/2} + W(u^n) \right) + D \rho_f^{n+\frac{1}{2}} &= \mu L (u_f^{n+\frac{1}{2}}) + p^{n+\frac{1}{2}}, \\
D \cdot u_f^{n+\frac{1}{2}} &= 0,
\end{align*}$$

(B)

$$\begin{align*}
\rho \frac{u_{s1}}{\Delta t/2} + D p_{s1} &= \mu L(u_{s1}), \\
D \cdot u_{s1} &= \psi_1(x) - \psi_0(x),
\end{align*}$$

(C)

$$\begin{align*}
\rho \frac{u_{s2}}{\Delta t/2} + D p_{s2} &= \mu L(u_{s2}), \\
D \cdot u_{s2} &= \psi_2(x) - \psi_0(x).
\end{align*}$$

The system (A), which involves the unknowns $u_f^{n+\frac{1}{2}}$ and $p_f^{n+\frac{1}{2}}$, is the discretization of the fluid equations in the absence of a source/sink. The system (B), whose unknowns are $u_{s1}$ and $p_{s1}$, involves a source/sink $\psi_1(x) - \psi_0(x)$ but does not include the time-dependent fluid density $p^{n+\frac{1}{2}}$ and the velocity fields $u^n$. Because the system (B) is time-independent, we only need to solve it for one time at the initial time step. The same technique is applied to the system (C) which has a source/sink $\psi_2(x) - \psi_0(x)$. Now by using the linearity of Eqs. (4.1)-(4.2) and the relation $Q_0 = -(Q_1 + Q_2)$, we can easily see that the solution $u^{n+\frac{1}{2}}$ and $p^{n+\frac{1}{2}}$ of Eqs. (4.1)-(4.2) can be written as $u^{n+\frac{1}{2}} = u_f^{n+\frac{1}{2}} + Q_1^{n+\frac{1}{2}} u_{s1} + Q_2^{n+\frac{1}{2}} u_{s2}$ and $p^{n+\frac{1}{2}} = p_f^{n+\frac{1}{2}} + Q_1^{n+\frac{1}{2}} p_{s1} + Q_2^{n+\frac{1}{2}} p_{s2}$, where $Q_1^{n+\frac{1}{2}}$ and $Q_2^{n+\frac{1}{2}}$ are source/sink fluxes in the regions $\Omega_1$ and $\Omega_2$, respectively.

The time-dependent flow in $\Omega_1$ is given by $Q_1(t) = \pi R_{up}^2 w_1(t)$, where $w_1(t)$ is the downward velocity in the center of the upstream of the vessel, see the right-top panel.
of Fig. 1. In order for the computed pressure level in Eq. (3.2) to be equal to the target blood pressure level \( \tilde{p}(t) \) which is shown in the right-bottom panel of Fig. 1, the following equation should hold

\[
\tilde{p}^{n+\frac{1}{2}} = \sum_x \left( p^{n+\frac{1}{2}}_f(x) + Q^{n+\frac{1}{2}}_1 p_{s1}(x) + Q^{n+\frac{1}{2}}_2 p_{s2}(x) \right) (\psi_2(x) - \psi_0(x)) h^{3}. \tag{4.11}
\]

From this relation, we can compute the flux \( Q^{n+\frac{1}{2}}_2 \) in the region \( \Omega_2 \), i.e.,

\[
Q^{n+\frac{1}{2}}_2 = \frac{\tilde{p}^{n+\frac{1}{2}} - \sum_x \left( p^{n+\frac{1}{2}}_f(x) + Q^{n+\frac{1}{2}}_1 p_{s1}(x) \right) (\psi_2(x) - \psi_0(x)) h^{3}}{\sum_x p_{s2}(x) (\psi_2(x) - \psi_0(x)) h^{3}}. \tag{4.12}
\]

Note that the prescribed flux \( Q_1 \) in the upstream region \( \Omega_1 \) and the given pressure level \( \tilde{p} \) in the downstream region \( \Omega_2 \) relative to the exterior region \( \Omega_0 \) uniquely determine the flux \( Q_2 \) in \( \Omega_2 \). The similar procedure to solve the fluid equations was also used in [3, 30, 46].

5 Results and discussion

First we discuss the method to derive the desired flow rate and blood pressure level in the vessel. As described in Section 3, the desired longitudinal velocity \( w_1(t) \) of the vessel is derived by generating the flux \( Q_1(t) \) in the region \( \Omega_{up} \), see Fig. 1. The top panel of Fig. 2 shows the simulated longitudinal velocity in the central region of the vessel (solid line) as a function of time, compared with the desired velocity \( w_1(t) \) (dashed line). We can see from the figure that the simulated longitudinal velocity is comparable to the desired velocity, which suggests that our method of driving a flow with the sink/source term enforces the velocity in the vessel to follow the experimental data well.

The blood pressure level, which is the interior pressure of the vessel relative to the exterior pressure, follows the compliance relation which says that the pressure level depends proportionally on the volume of the blood inside the artery. The desired (measured) pressure level \( \tilde{p}(t) \) is achieved by controlling the flux \( Q_2(t) \) in the region \( \Omega_2 \) and thus the volume inside the vessel, see Eq. (3.2) and Fig. 1. The middle panel of Fig. 2 compares the desired pressure level \( \tilde{p}(t) \) (dashed line) and the simulated pressure level (solid line) in the central region of the vessel. The latter reaches 112% of the former during the systolic phase.

The bottom panel of Fig. 2 shows the change of radius of the inner layer of the vessel at its center. The vessel radius is about 1.03 cm during the diastolic phase and 1.12 cm during the systolic phase. Thus the diameter changes about 8.7% from its diastolic minimum to its systolic maximum, which is similar to experimental data in literature, see [39]. Note that the overall patterns of the graphs in Fig. 2 indicate pulsatile dynamics.

The top panels of Fig. 3 show the velocity fields inside the vessel at three different times: 2.1 s (left), 2.2 s (middle), and 2.5 s (right). These times are indicated by ‘◦’ in
the top panel of Fig. 2. The velocity field in the middle figure is scaled by 120.0 cm/s, however, those in the other two figures are scaled by 12.0 cm/s. Therefore, the arrows in the middle figure indicates 10 times larger magnitude of the velocity than those in the other two figures. In the beginning of the systole at $t=2.1$ s, the blood in the central region away from the vessel wall begins to increase in the negative $z$ direction, while the blood flow is still vortical near the vessel wall. At the peak of the systolic phase at $t=2.2$ s, the flow is downward in the whole vessel domain. In the middle of the diastolic phase at $t=2.5$ s, the blood flow is again vortical, showing almost no unidirectional flow in the whole vessel domain.

The bottom panels of Fig. 3 show vorticity contours of the $x$ and $z$ components of the velocity field, $(u,w)$, at $t=2.1$ s (left), $t=2.2$ s (middle), and $t=2.5$ s (right). The velocity field $(u,w)$ is taken in the vertical plane $y=0$ which cuts the middle of the vessel. The vortical flows are observed near the vessel wall in the systolic phase at $t=2.1$ s and $t=2.2$ s with the region of the vortical flow being larger at the beginning of the systole.
at $t=2.1$ s than at the peak of the systolic phase at $t=2.2$ s. In the middle of the diastolic phase at $t=2.5$ s, the whole vessel domain is filled with the vortical flows.

Even though the pressure level and the radius of the artery exhibit qualitatively the same pulsatile behaviors as shown in Fig. 2, their behaviors are quantitatively different in real arteries. The arterial wall does not instantly respond to the pressure applied to the vessel wall; instead, the arterial wall lags behind the change of the pressure level, i.e., the \textit{hysteresis} occurs. It is known that the hysteresis appears when the damping coefficient is large enough [10, 30].

Fig. 4 shows the relationship between the pressure level ($x$-axis) and the radius of a vessel ($y$-axis) with various damping coefficients in Eq. (2.3). When the damping coefficient $c_r$ is zero, the arterial wall reacts immediately to the pressure level, and thus the relation between the radius and the pressure level is almost linear. As the damping co-
efficient increases, however, nonlinearity is pronounced, and a hysteresis loop appears. In the hysteresis loop, the radius of the vessel wall follows the pressure level with a time delay. As the damping coefficient increases further, the amount of time delayed in the motion of the artery relative to the pressure level increases as well, compare the area of the region enclosed by the loop in each panel of Fig. 4. Note that the loops in this figure are drawn in the counterclockwise direction in time. The hysteresis loop obtained from our simulations is similar to the experimental results observed from the measurements in canine aorta [1, 10] and in the human femoral artery [2, 10].

A mechanical and(or) a morphological change of an arterial wall may induce a change in the hemodynamics inside the wall, which then may result in arterial diseases such as aneurysm and stenosis, in which the arterial wall deforms into a certain shape. Aneurysm is a dilatation of a blood vessel, and a part of the arterial wall weakens and bulges outward. The aneurysm can occur in brain and coronary arteries, or at any part of the aorta; for example, thoracic aorta and descending aorta. Stenosis is an abnormal narrowing of a blood vessel which is mostly caused from atherosclerosis and results in a reduction in blood flow to the tissues. Although the exact pathogenesis of these diseases is not known,
previous studies suggest that the interplay between the changes in the hemodynamics of a blood vessel and the biological processes in the arterial vessel wall may be the main cause of the diseases \([6, 9, 14, 20, 21, 25, 32, 36, 39, 51, 54]\).

Our vessel model can be easily extended to investigate the blood flow dynamics in aneurysmal and stenotic arteries, since the IB method can handle arbitrary shapes of the immersed structure with elastic properties. Here we construct four different configurations of vessels related to the pathological deformations, which are symmetrically or asymmetrically dilated vessels, and symmetrically or asymmetrically constrictive vessels, see Fig. 5. The dilated vessels in both symmetric and asymmetric cases have the maximum radius of the bulging part to be around 2.5 cm which is approximately 250\% larger than the radius of a normal vessel shown in Fig. 1. The constrictive vessels have the minimum radius of the constrictive part to be around 0.6 cm (approximately 60\% of the radius of the normal vessel) for the symmetric case and 0.3 cm (approximately 30\% of the radius of the normal vessel) for the asymmetric case. In the simulations for these cases, we use the same data such as the downward velocity \(w_1(t)\), pressure level \(\bar{p}(t)\), and the physical and computational parameters as used before in the normal case.

Fig. 6 shows the velocity fields (top panels), vorticity contours (middle panels) of the velocity components, \(u\) and \(w\), and the trails of some fluid markers (bottom panels) for the case of the symmetrically bulging artery at three different times \(t = 2.1\) s (left), \(t = 2.2\) s (middle), and \(t = 2.5\) s (right). In the systolic phase (first and second columns), the blood flows down with vortices being confined in the bulging region. Especially, we can observe some separation lines between the vortical flow region and the mainstream flow region at the peak of the systolic phase at \(t = 2.2\) s (second column). In the diastolic phase (third column), the heart starts to relax after contraction and after the artery maximizes its
diameter, vortices leave the bulging region and a complicated flow pattern with almost no unidirectional flow appears in the whole vessel domain including the mainstream region. These observations become clearer in the bottom panels which draw the trails left by the fluid markers and which show their recent trajectories (streak lines).

The vortex dynamics in an aneurysmal artery is characterized by a sequence of the following different flow stages within one cardiac cycle: (i) flow separation and vortex formation at the proximal end in the mid-systole, (ii) downward translation of the vortex until the mid-diastole, and (iii) the most significant flow disturbance and disorganized recirculation until the onset of systolic acceleration in the next cardiac cycle. We can observe these typical flow stages from Fig. 6 which is comparable to Fig. 5 in [16] and Fig. 6 in [15].

In the case of an asymmetrically bulging artery, the flow field is similar to that of the symmetric bulging artery; there are clear separation lines between the vortical flow in the
bulging region and the downward flow in the mainstream region during systole, but the whole domain is filled with the complicated vortical flows during diastole, see Fig. 7 for the velocity fields (top), vorticity contours (middle) of the velocity components, $u$ and $w$, and the trails of some fluid markers (bottom) at three different times $t = 2.1$ s (left), $t = 2.2$ s (middle), and $t = 2.5$ s (right).

Fig. 8 shows the longitudinal velocity (top), the pressure level (middle) in the central region of the vessel, and the relative radius of the vessel wall (bottom) as functions of time for the cases of the symmetrically and asymmetrically dilated vessel, compared with those for the normal vessel case. The effective radius of the vessel can be computed as $\sqrt{A(t)/\pi}$, where $A(t)$ is the area of region enclosed by the cross-section of the inner layer of the vessel in the middle horizontal plane $z = 6.4$ cm. The bottom panel of Fig. 8 draws the effective radius of the vessel relative to the minimal effective radius computed at the end of the diastolic phase. It is shown that the overall behaviors of these three quantities for the dilated vessels are similar to those for the normal vessel except that the dilated
vessels have slightly slower flow speed (top) and slightly smaller relative radius of the vessel (bottom) than those of the normal vessel. This is because the vortical flows are confined in the bulging region and the flow in the mainstream region is not obstructed by the bulging part of the vessel.

This similarity between the bulging and normal vessels disappears in the case of stenotic arteries, in which the constrictive part of the vessel partially blocks the blood flow. Fig. 9 shows the longitudinal velocity (top), the pressure level (middle), and the relative radius of the vessel wall (bottom) for three different vessels: normal (solid line), symmetrically constrictive (dotted line), and asymmetrically constrictive (dashed line) arteries. The longitudinal velocity, the pressure level, and the relative radius of the vessels are similar for the normal artery and the asymmetrically constrictive artery. However, the magnitudes of these three quantities are much larger for the symmetrically constrictive artery than those for the normal artery. Since the downward flux through the vessel is almost the same regardless of the shape of the vessel, the flow speed should be larger when the flow passes through a smaller region. Clearly, the symmetrically constrictive artery runs down with a very narrow passage in the middle part of the vessel, in contrast to the normal artery or the asymmetrically constrictive artery, see Fig. 5.

Whereas the longitudinal velocity and the pressure level do not change much inside the dilated and normal vessels, those quantities go through a large change inside the symmetrically constrictive vessel. Fig. 10 shows comparison of the longitudinal velocity...
Figure 9: Comparison of the longitudinal velocity (top), the pressure level (middle), and the relative radius of the vessel wall (bottom) for three different shapes of the vessel: normal (solid line), symmetrically constrictive (dotted line), and asymmetrically constrictive (dashed line) arteries.

(upper-right panel) and the pressure level (lower-right panel) in three regions inside the symmetrically constrictive vessel. The three regions are chosen in the upstream region ($R_1$), the middle narrowest region ($R_2$), and the downstream region ($R_3$), as indicated in the left panel of Fig. 10. In the upstream region $R_1$, it is observed that the flow speed is very low and the pressure level is very high. As the blood passes through the narrow region $R_2$ and goes further down to the downstream region $R_3$, the flow speed increases substantially and the pressure drops rapidly.

Fig. 11 shows the velocity fields (first and third rows) and the trails of fluid markers (second and fourth rows) for the case of the symmetrically (upper two rows) and the asymmetrically (lower two rows) constrictive arteries at three different times: 2.1 s (left), 2.2 s (middle), and 2.5 s (right). In the systolic phase at $t=2.1$ s and 2.2 s, the vortical flows are confined in the vicinity of the vessel wall and in the region behind the constricted part, while the flow moves downward in the mainstream region for both symmetrically and asymmetrically constrictive arteries. In the diastolic phase at $t = 2.5$ s, however, the whole domain including the front of the constricted part is filled with complicated vortical flows.

The vortex dynamics in a stenotic vessel shows typical flow stages during one cardiac cycle as follows. During the systolic phase, a vortex is formed near the wall distal to the stenosis of the blood vessel and grows to build a recirculation region at the post-stenotic part. We can also observe recirculation flows at distal and proximal regions to the stenosis.
in the mid-diastole. These recirculation flows almost disappear before the onset of the next systolic acceleration. These typical flow stages are observed both in our simulation results (Fig. 11) and in [6, 26, 52].

6 Summary and conclusions

We have developed a three-dimensional mathematical model to study the interaction between blood flow and the aortic wall. Since the pulsatile flow affects the aortic wall movement which in turn influences the flow pattern, we need to consider a coupled system for the fluid-structure interaction. In computer simulations which are based on the IB method, we prescribed the fluid velocity in the upstream using the MR data obtained from a healthy subject and the pressure level in the downstream using the transmural pressure reproduced from literature. The source/sink term which we used in our model has been shown to drive blood flow in the vessel while satisfying well the desirable downward velocity and transmural pressure level.

Blood vessels exhibit nonlinearity in material and mechanical properties. It is therefore important for a model to account for these nonlinear properties in order to make reliable predictions for the subjects. The hysteresis behavior, which is one of the key characteristics of the blood vessel properties, has been found in the relation between the
blood pressure and the diameter of a vessel, when the viscoelastic property of the arterial wall is taken into consideration in the model as reported in literature.
It is commonly believed that arterial diseases such as aneurysm and stenosis may result from a combination of change in hemodynamics of blood flow and morphological change of the vessel wall due to abnormal biological processes. Therefore, we have considered five different configurations of vessels: a straight but tapered artery as a normal vessel, a symmetrically and asymmetrically dilated arteries as aneurysmal vessels, and symmetrically and asymmetrically constrictive arteries as stenotic vessels, and investigated how the shape of arteries influences the flow patterns and pressure distributions inside the vessel. In the aneurysmal arteries, flow speed and relative radius of the vessel are slightly reduced as the degree of dilation is increased, but the pressure level maintains the same. In the stenotic arteries, however, there are drastic changes in flow velocity, pressure level, and the relative radius. In general, a narrower vessel produces higher flow speed and higher pressure level, which pushes the vessel wall more outward, resulting in higher relative radius.

The primary goal of this research is to develop a computational model that can be applied to investigate the hemodynamics of healthy human subjects or patients who may have aneurysms or stenosis. The IB method enables us to construct an artery model based on a real anatomy of a subject and to incorporate experimental data into the model. This computational model along with the data analysis may provide a tool to envision potential causes of aneurysm or stenosis and a guidance for optimal treatment to improve the effect of various stents or to prevent a possible rupture.

Acknowledgments

Lim was supported by Charles Phelps Taft Research Center at University of Cincinnati, Kim by the Chung-Ang University research grant in 2015 and by National Research Foundation of Korea Grant (Grant No. 2015R1A2A2A01005420), and Park by the Chung-Ang University Excellent Student Scholarship in 2013.

References


