An Image-Based Computational Fluid Dynamic Method for Haemodynamic Simulation*

Nami MATSUNAGA**, Hao LIU***
and Ryutaro HIMENO***

Blood vessel in circulatory system often shows a rich variety in geometry as well as in morphology and image-based anatomic modeling of such vessels for haemodynamic simulation is usually of great time-consumption. In this paper we proposed a new computational fluid dynamic method that is capable to directly utilize the pixel and/or voxel dataset based on the threshold for the raw medical images of MRI, CT and etc. to define the computational domain in a Cartesian coordinate system. Boundary of the domain is determined in a manner of VOF (fractional Volume Of Fluid) and the flow is computed using a non-staggered finite difference method, in which treatment of the boundary conditions is conducted with the neighboring point local collocation method (NPLC). Results are presented of two pulsatile flows in a two-dimensional stenosed blood vessel model. Comparison with other reliable results shows that the present method is able to reasonably predict complicated vortical flow in blood vessel very well.

Key Words: Biomechanics, Image Processing, Modeling, Navier–Stokes Equation, Cartesian Coordinate System, Finite Difference Method, Stenosis, Blood Vessel

1. Introduction

Blood vessel in circulatory system often shows a rich variety in geometry as well as in morphology and such richness & complexity of the geometry of vessels as with rapid change in cross-sectional shape or area (stenosis), curvature or torsion would always harbor complex, separated flow patterns. This haemodynamic feature enhances difficulties to understand the detailed flowfields with applications to wall shear stress (WSS), a major factor in the onset, the development, and the outcome of the arteriosclerosis (Fry et al.10; Caro9; Ku et al.9) and makes it very difficult to predict flow and hence the WSS from a previous knowledge of another. Hence, anatomically realistic modeling of blood vessel on a basis of medical images is of great importance in computation of blood flow, in particular with consideration of the individual–specific simulation of haemodynamics for clinical purpose. Such image–based modeling of blood vessel is, however, often of great time-consumption. Because the conventional computational fluid dynamic methodology, in terms of finite element method, or finite difference method, or finite volume method, is mostly established in the so-called BFC (Boundary–Fitted Coordinate) system, that often requires tremendous efforts in grid generation about smoothened, 3D geometry of the ROI (Region Of Interest) that is generally extracted from unclear raw medical images.

In this study, we notice that, given a set of medical images, the outline of an object can be defined roughly by means of the threshold for the raw images.
and thus boundaries of the object can be described in a manner of VOF (Fractional Volume Of Fluid) that is defined from the color information of each pixel or voxel obtained from the MRI pictures. It is supposed that some noises involved in the original images have been removed to a certain extent. The computational domain can then be discretized simply in a Cartesian coordinate system. However, treatment of boundary conditions at such VOF-based boundary stencils turns out to be quite difficult. In this paper we proposed a new computational method that is capable to directly utilize the pixel and/or voxel dataset and to define the computational domain in a Cartesian coordinate system in a manner of VOF and the flow is computed using a non-staggered finite difference method, in which treatment of the boundary conditions is conducted with the Neighboring Point Local Collocation method (NPLC)\textsuperscript{(45)}. A method by Hirt et al.\textsuperscript{(46)} was introduced to search and determine the boundary on a basis of the voxel information. Pressure conditions at boundary stencils are defined using the Neumann conditions being combined with the equation of continuity. In the following, we first give a detailed description of the method and then present results of two pulsatile flows in a two-dimensional stenosed blood vessel model. Comparison with other reliable results of Liu et al.\textsuperscript{(47)} shows that the present method is able to reasonably predict complicated vortical flow in blood vessel very well.

2. Definition of Geometric Model and Inflow Flow

Consider the following two-dimensional stenosed channel model\textsuperscript{(47-49)} as depicted in Fig. 1.

The upper wall is described by the following function

\[ F(x) = \begin{cases} 1 & (x_1 < x < x_2), \\
1 - 0.5e^{1 + \tan(h(x - x_3))} & (x_2 < x < x_3), \\
1 - e & (x_3 < x < x_4), \\
1 - 0.5e^{1 - \tan(h(x - x_4))} & (x_4 < x < x_5), \\
1 & (x_5 < x < x_6). 
\end{cases} \]

where

\[ x_1(x_3 + x_0)/2, \quad x_2 = (x_4 + x_5)/2, \]
\[ a \text{ is the slope parameters, } x_i (i = 1, \ldots, 6) \text{ are constants, and } e \text{ is the height of the stenoses}. \]

At inlet the waveforms of two pulsatile flows with a mean velocity of \( U_0 \) as shown in Fig. 2 are defined as

\[ U_0(t) = \begin{cases} 0.5(1 - \cos(2\pi t)) & (\text{Sinusoidal}), \\
0.251 + 0.290(\cos \varphi + 0.97 \cos 2\varphi) & (\text{Non-sinusoidal}), \\
+0.473 \cos 3\varphi + 0.14 \cos 4\varphi \end{cases} \]

where \( \varphi = 2\pi t - 0.14142 \). The points \( a, b, c, d, e \) and \( f \) in Fig. 2 denote the points where \( \partial U_0/\partial t = 0 \) and \( \partial^2 U_0/\partial t^2 = 0 \), respectively. The nonsinusoidal pulsatile flow is achieved by introducing a physiological volumetric flow as the general form given by McDonald\textsuperscript{(10)}.

3. Determination of the Boundary from Voxel Information

The color information of each voxel based on MRI images can be described as voxel data in a manner of the Volume of Function (VOF), where \( V \) varies over a range of \( 0 \leq V \leq 1 \). The \( V \) is also regarded as the volume of the fluid in each voxel, which implies that a boundary exists inside the voxel with a value of \( 0 < V < 1 \). We then determine the boundary of a domain as given in the following way.

First, suppose that a rectangular domain \( \Omega \) covering the stenosed model \( \Omega_1 \) as in Fig. 1 is given and the voxel data are defined with a grid spacing of \( h=\Delta x \) and \( k=\Delta y \) in \( x \) and \( y \)-directions for \( \Omega \), respectively. Here, grid points are basically located at the center of each voxel or cell. The four grid points of the cell denoted by \( P_x, P_y, P_z, P_e \) are adjacent to \( P \in \Omega_1 \), and their distances to \( P \) are denoted by \( h_x, h_y, h_z, h_e \) (cf. Fig. 3). Note that these four points are not necessary to be located at the center of the voxel or cell. For example, if \( V(P_e) = 1 \) and \( V(P_{ew}) = 1 \), the spatial coordinate of the point \( P_e \) corresponding to one of the boundary points as illustrated in Fig. 3 is defined such that

\[ (x_e, y_e) = (ih + h_x, jh) \]

where \( h_x = (0.5 + V(P_e))h \), which leads to \( 0.5h < h_x < 1.5h \).
4. Numerical Methods

The two-dimensional, unsteady, incompressible Navier-Stokes equations are written as,

\[ \frac{\partial u}{\partial t} + \frac{\partial}{\partial x}(u^2) + \frac{\partial}{\partial y}(uv) = -\frac{\partial p}{\partial x} + \frac{1}{Re} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \]
\[ \frac{\partial v}{\partial t} + \frac{\partial}{\partial x}(uv) + \frac{\partial}{\partial y}(v^2) = -\frac{\partial p}{\partial y} + \frac{1}{Re} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right), \]

where \((u, v)\) is velocity components, \(p\) is pressure, \(St\) is Strouhal number, \(F_x\) and \(F_y\) are flux terms being defined as,

\[ F_x := -u \frac{\partial u}{\partial x} - v \frac{\partial u}{\partial y} + \frac{1}{Re} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \]
\[ F_y := -u \frac{\partial v}{\partial x} - v \frac{\partial v}{\partial y} + \frac{1}{Re} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right), \]

where \(Re\) is Reynolds number. Note that \(Re\) and \(St\) are defined as \(Re = U_p/\nu\) and \(St = d^3/U_pT\) respectively, where \(U_p\) is the peak volumetric flow rate per unit depth of the channel; \(\nu\) is kinematic viscosity; \(d\) is the unperturbed width of the channel; and \(T\) is the period of a complete cycle. Note that the volume of fluid at each voxel as shown in Fig. 4 should be calculated in advance.

In addition, a cell-vertex architecture for velocity \((u, v)\) and pressure \(p\) is employed and a non-staggered finite difference method by Nishida(11) is utilized as well as for the spatial discretization. However, since this method requires ideal points for calculation of the differential terms at the boundary, the Neighboring Point Local Collocation method (NPLC) by Nakano et al.(4,13) is hereby introduced at near the boundary. To solve Eqs. (1) and (2), we use the method of line approach, which decouples the spatial discretization from the temporal discretization. Furthermore, since the divergence free condition is satisfied at each time step, the coupling form of velocity and pressure is used. Details of algorithm are given as below.

• Algorithm

Step 1. Time integration is discretized using the Adams-Bashforth method with a secondary accuracy for flux terms, and the explicit Euler method is employed for other terms.

The transient velocity \((u^*, v^*)\) at time step of \(n+1\) is as,

\[ u_{i+\frac{1}{2},j}^{n+1} = u_{i+\frac{1}{2},j}^n \]
\[ + \frac{\Delta t}{St} \left( \frac{3F_x^{i+\frac{1}{2},j-\frac{1}{2}} - 4F_x^{i+\frac{1}{2},j} + F_x^{i+\frac{1}{2},j+\frac{1}{2}}}{2} \right) \]
\[ + \frac{\Delta t}{St} \left( \frac{3F_y^{i+\frac{1}{2},j+\frac{1}{2}} - 4F_y^{i+\frac{1}{2},j} + F_y^{i+\frac{1}{2},j-\frac{1}{2}}}{2} \right) \]

where superscript \(n\) denotes the \(n\)th time step, \(\Delta t\) is time step, and \(i, j\) are grid points in \(x\) and \(y\) direction, respectively. Here, we put \(F_x^{i+\frac{1}{2},j+\frac{1}{2}} := F_x^{i+1,j+2}\) and \(F_y^{i+\frac{1}{2},j+\frac{1}{2}} := F_y^{i+1,j+2}\).

Step 2. To compute pressure \(p_{i+\frac{1}{2},j}^{n+1}\), we have

\[ \frac{\partial p^{n+1}}{\partial x} \bigg|_{i+\frac{1}{2},j} = -\frac{\partial u^{n+1}}{\partial x} \bigg|_{i+\frac{1}{2},j} - \frac{\partial v^{n+1}}{\partial y} \bigg|_{i+\frac{1}{2},j}, \]
\[ \frac{\partial p^{n+1}}{\partial y} \bigg|_{i+\frac{1}{2},j} = -\frac{\partial u^{n+1}}{\partial x} \bigg|_{i+\frac{1}{2},j} - \frac{\partial v^{n+1}}{\partial y} \bigg|_{i+\frac{1}{2},j}, \]

and

\[ D_{i+\frac{1}{2},j}^{n+1} = 0. \]

Then, substituting Eqs. (5) and (6) into Eq. (7) a pressure-based Poisson equation is gained such that,

\[ \frac{\partial^2 p^{n+1}}{\partial x^2} \bigg|_{i+\frac{1}{2},j} + \frac{\partial^2 p^{n+1}}{\partial y^2} \bigg|_{i+\frac{1}{2},j} = \frac{1}{St} \Delta t \frac{\partial^2 u}{\partial x^2} \bigg|_{i+\frac{1}{2},j} + \frac{\partial^2 v}{\partial y^2} \bigg|_{i+\frac{1}{2},j} \]

Step 3. From Eqs. (5) and (6) we can calculate a velocity at time step \(n+1\). If \(\left| D_{i+\frac{1}{2},j}^{n+1} \right| < \epsilon^*, \)

where \(\epsilon^*\) is a small constant, then do updating as,

\[ u_{i+\frac{1}{2},j}^{n+1} = u_{i+\frac{1}{2},j}^n, \quad v_{i+\frac{1}{2},j}^{n+1} = v_{i+\frac{1}{2},j}^n, \quad p_{i+\frac{1}{2},j}^{n+1} = \rho_{i+\frac{1}{2},j}^n, \]

and return to Step 2. Otherwise, go to Step 1.

Note that the flux terms \(F_x\) and \(F_y\) at the midpoint of each cell side need to be calculated in Step 1. In order to calculate these terms, the non-staggered finite difference method introduced by Nishida(11) is used basically. The method is as follows.

The velocity at each cell center is calculated
using the mean value of two adjacent grid points.
\[
\begin{align*}
\frac{u_{i+\frac{1}{2},j} - u_{i-\frac{1}{2},j}}{2} &= \frac{u_{i,j} + u_{i+1,j}}{2}, \\
\frac{v_{i,j+\frac{1}{2}} - v_{i,j-\frac{1}{2}}}{2} &= \frac{v_{i,j} + v_{i,j+1}}{2}.
\end{align*}
\] (8)

Hence, we have
\[
\begin{align*}
\frac{\partial u}{\partial x} \mid_{i+\frac{1}{2},j} &\approx \frac{u_{i+1,j} - u_{i,j}}{h}, \\
\frac{\partial v}{\partial y} \mid_{i,j+\frac{1}{2}} &\approx \frac{1}{2} \left( \frac{u_{i+1,j+1} - u_{i+1,j-1} + u_{i,j+1} - u_{i,j-1}}{2k} \right), \\
\frac{\partial^2 u}{\partial x \partial y} \mid_{i+\frac{1}{2},j} &\approx \frac{1}{2k} \left( \frac{u_{i+2,j} - u_{i-1,j} - u_{i,j} - u_{i-1,j-1}}{h} \right), \\
\frac{\partial^2 u}{\partial y^2} \mid_{i,j+\frac{1}{2}} &\approx \frac{1}{2k} \left( \frac{u_{i+1,j+2} - 2u_{i+1,j+1} - u_{i,j+1}}{h} \right) + \frac{u_{i,j+1} - 2u_{i,j} + u_{i,j} - 1}{k^2}.
\end{align*}
\] (9)

for the discretization of flux term \(F_{x,i+j+1/2} \). Similarly, we can also obtain
\[
\begin{align*}
\frac{\partial y}{\partial x} \mid_{i+\frac{1}{2},j} &\approx \frac{1}{2k} \left( \frac{v_{i+2,j} - v_{i-1,j} - v_{i,j} - v_{i-1,j-1}}{h} \right), \\
\frac{\partial y}{\partial x} \mid_{i,j+\frac{1}{2}} &\approx \frac{1}{2k} \left( \frac{v_{i+1,j+2} - 2v_{i+1,j+1} - v_{i,j+1}}{h} \right) + \frac{v_{i,j+1} - 2v_{i,j} + v_{i,j} - 1}{k^2}.
\end{align*}
\] (10)

for the flux term \(F_{y,i+j+1/2} \). In discretization of the term \(\partial p / \partial x \), we employ
\[
\begin{align*}
\frac{\partial p}{\partial x} \mid_{i+\frac{1}{2},j} &\approx \frac{p_{i+1,j} - p_{i,j}}{h}, \\
\frac{\partial p}{\partial y} \mid_{i,j+\frac{1}{2}} &\approx \frac{p_{i,j+1} - p_{i,j}}{k}.
\end{align*}
\] (11) (12)

In case of \(V(P_{i,j})=1 \) and \(V(P_{i+1,j})=0 \), we replace Eq. (11) with
\[
\begin{align*}
\frac{\partial p}{\partial x} \mid_{i+\frac{1}{2},j} &\approx \frac{p(P_{i+1,j}) - p_{i,j}}{h_E}.
\end{align*}
\] (13)

Furthermore, the differential terms at neighboring points \((i+1/2, j) \) or \((i, j+1/2) \) at the boundary are calculated using the Neighboring Point Local Collocation method (NPLC) methods in terms of flux terms so as to avoid use of ideal points. With the local collocation method, the two-dimensional spatial derivatives at boundary stencils are approximated as below.

First, we number the points as shown in Fig. 5 and assume that a flow variable is approximated by
\[
\begin{align*}
f(x, y) &= f_0 + a_1 x + a_2 y + a_3 x^2 + a_4 y^2 + a_5 x y,
\end{align*}
\] (14)

where \(f_0 \) represents the value at the current point, \(a_i \) \((i = 1, \ldots, 5) \) are unknown coefficients, and \((x, y) \) is a relative coordinate from the current point. Note that we can obtain the following spatial derivatives at point 0 as shown in Fig. 5.
\[
\begin{align*}
\frac{\partial f}{\partial x} &\approx a_1 + 2a_3 x + a_5 y, \\
\frac{\partial f}{\partial y} &\approx a_2 + 2a_4 y + a_5 x, \\
\frac{\partial^2 f}{\partial x^2} &\approx 2a_3, \\
\frac{\partial^2 f}{\partial y^2} &\approx 2a_4,
\end{align*}
\]

by differentiating Eq.(14) with respect to \(x \) or \(y \). To get unknown coefficients \(a_i \) \((i = 1, \ldots, 5) \), the following matrices for \(i = 5, \ldots, 8 \) are solved and averaged to ensure the spatial second-order accuracy.
\[
\begin{align*}
(x_1, y_1, x_1^2, y_1^2, x_1 y_1, a_1) &= f_1 - f_0, \\
x_2, y_2, x_2^2, y_2^2, x_2 y_2, a_2 &= f_2 - f_0, \\
x_3, y_3, x_3^2, y_3^2, x_3 y_3, a_3 &= f_3 - f_0, \\
x_4, y_4, x_4^2, y_4^2, x_4 y_4, a_4 &= f_4 - f_0, \\
x_5, y_5, x_5^2, y_5^2, x_5 y_5, a_5 &= f_5 - f_0.
\end{align*}
\]

Here, if a grid point in the diagonal direction (for example, the point 6 in Fig. 5) is not in the region \(\Omega \), the calculation of the matrix is skipped.

For the convection terms we use the third-order upwind scheme at usual inner grids and an alternative upwind scheme with the first-order by means of the NPLC method at boundary. Note that we further introduce the following method to assess the convection terms at points \((i+1/2, j) \) and \((i, j+1/2) \). For example, \(u \cdot \frac{\partial u}{\partial x} \), one of the convection terms \(F_x \) at point \(P_{i, j} \) is calculated with the third-order upwind scheme such as,
\[
\begin{align*}
\frac{u \cdot \frac{\partial u}{\partial x} \mid_{i+\frac{1}{2},j}}{h} &\approx \frac{\left( u_{i+\frac{1}{2},j} \cdot 2d_{18} + 3d_{9} - 6d_{5} + d_{3} \right) - \left( u_{i+\frac{1}{2},j} \cdot 2d_{18} + 3d_{9} - 6d_{5} + d_{3} \right)}{6h} & u_{i+\frac{1}{2},j} > 0, \\
\frac{u \cdot \frac{\partial u}{\partial x} \mid_{i+\frac{1}{2},j}}{h} &\approx \frac{\left( u_{i+\frac{1}{2},j} \cdot 2d_{18} + 3d_{9} - 6d_{5} + d_{3} \right) - \left( u_{i+\frac{1}{2},j} \cdot 2d_{18} + 3d_{9} - 6d_{5} + d_{3} \right)}{6h} & u_{i+\frac{1}{2},j} \leq 0,
\end{align*}
\] (15)

where
\[
\begin{align*}
d_{9} &= \frac{u_{i+1,j} + u_{i-1,j}}{2}, \\
d_{1} &= \frac{u_{i,j+1} + u_{i,j-1}}{2}, \\
d_{0} &= \frac{u_{i+1,j} + u_{i,j+1} + u_{i,j-1} + u_{i-1,j}}{4}, \\
d_{1} &= \frac{u_{i+1,j} + u_{i,j+1} + u_{i,j-1} + u_{i-1,j}}{4}, \\
d_{8} &= \frac{u_{i+1,j} + u_{i,j+1} + u_{i,j-1} + u_{i-1,j}}{4},
\end{align*}
\]

if all referring to grid points are included to the region \(\Omega \). The points \(\circ \) in Fig. 6 denotes the grid points to be referred and the point \(\bullet \) is the point to be calculated. Note that accuracy of the local truncation error in the convection terms turns out to be the second-order.
because of the second-order accuracy in the calculation of \(d\). Otherwise, we calculate the terms using the first and second differentiable terms by the NPLC method and the following equation

\[
\frac{\partial u}{\partial x} \bigg|_{i-\frac{1}{2}, j} \approx c \frac{u_{i+\frac{1}{2}, j} - u_{i-\frac{1}{2}, j}}{h},
\]

where, \(c\) is the value of the first and second differentiation obtained from NPLC, respectively. Note that if point \(P_{i+1/2, j}\) is close to the boundary the upwind scheme is reduced to the first order because of the usage of the NPLC method.

Similarly, the convection terms in \(y\) direction can be obtained.

5. Boundary Conditions

In the solutions to the Navier-Stokes equations with Dirichlet conditions we usually need the Neumann conditions for the pressure. However, it is very difficult to treat the boundary conditions for an arbitrary region except a square or a rectangle in the Cartesian coordinate system. We hereby propose a new method\(^{12}\) that is based on Eq. (1) for this problem. Let \(n=(n_x, n_y)\) be the unit outer normal vector. When the non-slip condition in the boundary is imposed, it holds

\[
\frac{\partial p}{\partial n} = \frac{\partial p}{\partial x} n_x + \frac{\partial p}{\partial y} n_y = 0
\]

(17)

for the pressure. Moreover, if \(n_x \neq 0\) and \(n_y \neq 0\), we have

\[
\frac{\partial u}{\partial t} + u \cdot \nabla u + \nabla p = \frac{\partial}{\partial x} \left( F \frac{\partial u}{\partial x} + S \right),
\]

(18)

\[
\frac{\partial v}{\partial t} + v \cdot \nabla u + \nabla p = \frac{\partial}{\partial y} \left( F \frac{\partial v}{\partial y} + S \right),
\]

(19)

by multiplying both sides of the Eq. (1) with \(n_x\) and \(n_y\), respectively, and we obtain

\[
\frac{\partial p}{\partial x} = h \left( F \frac{\partial u}{\partial x} + S \right) - n_x \frac{\partial v}{\partial y} + S
\]

(20)

from Eqs. (17), (18) and (19). Therefore, if \(P_k\) is a boundary point on the right side of the point \(P_{i,j}\), we can calculate the \(p(P_k)\) by using

\[
\frac{\partial p}{\partial x} = \frac{p(P_k) - p(P_{i,j})}{h},
\]

where \(p(P_k)\) and \(p(P_{i,j})\) denote the pressures at points of \(P_k\) and \(P_{i,j}\), and \(h\) is the length between \(P_k\) and \(P_{i,j}\). In case of that \(n_x=1\) and \(n_y=0\), we may use

\[
\frac{\partial p}{\partial x} = F_s - \frac{\partial u}{\partial t} St.
\]

6. Results and Discussions

Computations were conducted at \(Re = 750\), \(St = 0.024\) with a stenosed channel of \(e=0.5\). A grid system with a uniform grid spacing of \(h=k=0.05\) is used for all the cases. The time step is set to be \(\Delta t = 1.25 \times 10^{-4}\) and the computations were carried out up to 8000 steps. At solid wall of the channel we apply the non-slip condition for the velocity components:

\[
(u, v) = (0, 0).
\]

The inflow conditions are taken

\[
(u, v) = (U_0, 0).
\]

For the outer flow conditions we use

\[
\frac{\partial u}{\partial x} = 0, \quad \frac{\partial v}{\partial x} = 0, \quad \frac{\partial U_0}{\partial t} St = \frac{\partial p}{\partial x}.
\]

The initial condition is that the flow is at rest at \(t = 0\). Poisson equations for pressure is solved using the SOR method with the acceleration parameter \(\omega = 1\) and the stopping criterion for the iteration was

\[
\sum_{P \in E} \frac{|p_{P}^{n+1} - p_{P}^{n} - p_{P}^{n}|}{|p_{P}^{n}|} < 10^{-4}.
\]

Moreover, we used

\[
\sum_{P \in E} \frac{|D_{P}^{n+1} - D_{P}^{n}|}{|D_{P}^{n}|} < 10^{-4}
\]

and

\[
\max_{P \in E} |D_{P}^{n+1}| < 10^{-2}
\]

as the stop criterion for the divergence free condition \(D_{P}^{n+1} = 0\) in step 3. The Wall Shear Stress (WSS : \(\tau_w\)) on the lower wall is defined as the dimensionless shear stress on the wall surface such that \(\tau_w = (\partial u / \partial y)_{wall} / Re\).

The WSS (Wall Shear Stress) distributions on lower walls were plotted in Figs. 7 – 10. Figures 11 – 14 show iso-velocity and pressure contours in both sinusoidal and nonsinusoidal cases.

Figures 7 – 10 show that higher stress is observed immediately behind the stenosed portion when the pulsatile flow just over the peak in both cases. In Figs. 8 and 10 we compared the present results with those of Liu et al\(^{(18)}\) at three specific instants. Although slight discrepancy is observed somewhere in the distribution, it is seen that the present method can catch the feature of such a complex vortical flow reasonably well. Moreover, as shown in Figs. 11 – 14, when the flow turns to deceleration, the wavy core flow behind the
Fig. 7 Wall shear stress distribution at $Re=750$, $St=0.024$ with $\varepsilon=0.5$ in sinusoidal case

(b) Liu et al. (1999)

Fig. 8 Comparison of wall shear stress distribution with those by Liu et al. (1999) in sinusoidal case

Fig. 9 Wall shear stress distribution at $Re=750$, $St=0.024$ with $\varepsilon=0.5$ in nonsinusoidal case

(b) Liu et al. (1999)

Fig. 10 Comparison of wall shear stress distribution with those by Liu et al. (1999) in nonsinusoidal case
Fig. 11 Iso-velocity contours at $Re=750, Sl=0.024$ with $\varepsilon=0.5$ in sinusoidal case

(b) Liu et al. (1999)

Fig. 13 Iso-velocity contours at $Re=750, Sl=0.024$ with $\varepsilon=0.5$ in nonsinusoidal case

(b) Liu et al. (2001)

Fig. 12 Pressure contours at $Re=750, Sl=0.024$ with $\varepsilon=0.5$ in sinusoidal case

(b) Liu et al. (1999)

Fig. 14 Pressure contours at $Re=750, Sl=0.024$ with $\varepsilon=0.5$ in nonsinusoidal case

(b) Liu et al. (2001)
stenoses, gradually developing to strong vortices as detected in Liu et al. Since the inflow velocity changes rapidly in the case of nonsinusoidal waveform, the vortices are stronger than the case of sinusoidal waveform. Overall, we believe the vortex wave flow is dominated by not only the stenoses geometry but also the waveform of the inflow. Such phenomenon is also observed experimental and numerical studies by Sobey[13] and Tutty et al.[14]

7. Conclusions

The present study demonstrates the feasibility of medical image-based computational fluid dynamic modeling of complex pulsatile, vortical blood flow directly in a Cartesian coordinates system. This method shows great potential in definition of the boundary based on the voxel information as well as grid generation in case of complex geometry. Quantitative comparison with other reliable results further confirms that our method can reasonably predict not only vortical flow patterns but also wall shear stress distributions. Currently, application of simulation of blood flows in a carotid artery based on MR images is undergoing and an extension to the three-dimensional case is also under construction.

References