International Journal of Medical, Medicine and Health Sciences
ISSN: 2517-9969
Vol:3, No:1, 2009

Blood Cell Dynamics in a Simple Shear Flow using an Implicit Fluid-Structure Interaction Method Based on the ALE Approach

Choeng-Ryul Choi, Chang-Nyung Kim, and Tae-Hyub Hong

***Abstract*—**A numerical method is developed for simulating the motion of particles with arbitrary shapes in an effectively infinite or bounded viscous flow. The particle translational and angular motions are numerically investigated using a fluid-structure interaction (FSI) method based on the Arbitrary-Lagrangian-Eulerian (ALE) approach and the dynamic mesh method (smoothing and remeshing) in FLUENT (ANSYS Inc., USA). Also, the effects of arbitrary shapes on the dynamics are studied using the FSI method which could be applied to the motions and deformations of a single blood cell and multiple blood cells, and the primary thrombogenesis caused by platelet aggregation. It is expected that, combined with a sophisticated large-scale computational technique, the simulation method will be useful for understanding the overall properties of blood flow from blood cellular level (microscopic) to the resulting rheological properties of blood as a mass (macroscopic).

***Keywords*—**Blood Flow, Fluid-Structure Interaction (FSI), Micro-Channels, Arbitrary Shapes, Red Blood Cells (RBCs)

I. Introduction

In the microcirculation, the flow behavior of RBCs plays a crucial role in many physiological and pathological phenomena. For example, the random-like transverse motion and rotation of RBCs in shear flow is believed to play an important role in thrombogenesis. However, the role of RBCs in the mass transport mechanism of cells and proteins to the thrombus is still not completely understood [1-3]. As a consequence, many studies have been performed on both the rheological and microrheological behavior of RBCs flowing through capillaries [4-7]. The growing interest in the modeling and simulation of biomedical systems and, in particular, the human cardiovascular system, is supported by the numerous works [8-11]. Within this context, the application of mathematical and numerical models has been shown to provide useful information. Detailed knowledge on the motion of RBCs flowing in micro-channels under simple shear flow and the influence of blood flow is essential to provide a better understanding on the blood rheological properties and blood cell aggregation.

All authors are with the Department of Mechanical Engineering, College of Advanced Technology, Kyunghee University, Republic of Korea (†corresponding author to provide phone: 82-31-201-2869; fax: 82-31-202-9715; e-mail: choicr@gmail.com).

The objectives of this study are to develop for simulating the motion of particles with arbitrary shapes in an effectively infinite or bounded viscous flow and to investigate the transverse motion and rotation of particles with arbitrary shapes suspending in a simple shear flow designed with two parallel plates. The arbitrary shapes are assumed to be the following shapes: disk, spherical, ellipsoidal and biconcave shapes. The effects of the shapes on the transverse motion and rotation of the particles are estimated. In order to take the interaction between fluid and the particles into account, a fluid-structure interaction (FSI) method based on the Arbitrary-Lagrangian-Eulerian (ALE) approach and the dynamic mesh method (smoothing and remeshing) in FLUENT (ANSYS Inc., USA) is organized.

II. Materials and Methods

A. Definition of the Analysis Models

In order to investigate the effects of the shapes of particles with arbitrary shapes on the transverse motion and rotation of the particles, several shapes are considered; disk, spherical, ellipsoidal and biconcave shapes. The radius of the spherical shape is 3.07 mm, the lengths of the semimajor and semiminor axes of the ellipsoidal shape are 4.87 mm and 2.43 mm, respectively. The biconcave shape (representing RBC) is modeled with the three-dimensional surface formula of

$$x = R\sqrt{1-(y^2+z^2)/R^2}[c_0 + c_1(y^2+z^2)/R^2 + c_2(y^2+z^2)/R^4]$$

where R, c0, c1 and c2 are 3.91 mm, 0.1035805, 1.001279, -0.561381, respectively. The radius of the shape is 3.91 mm and the thickness is 3.384 mm. The volumes of the three shape particles are the same.

We consider two types of the flow channel designed with two parallel plates as shown in Fig. 1 and Fig. 2. The upper plate moves at a constant velocity and the bottom one is fixed (Fig. 1: MODEL 1) or moves at a constant (Fig.2: MODEL 2). The distance between the two plates is 20 mm. The present shear rates (du/dy) of the simple shear flows range from 1 to 32 s^{-1}. The density of the fluid (plasma) is 1,060 kg/m^3 and its viscosity is 3.5 cp.

B. Numerical Method

The mesh generation is performed by using software GAMBIT (ANSYS Inc., USA). The simulations are performed

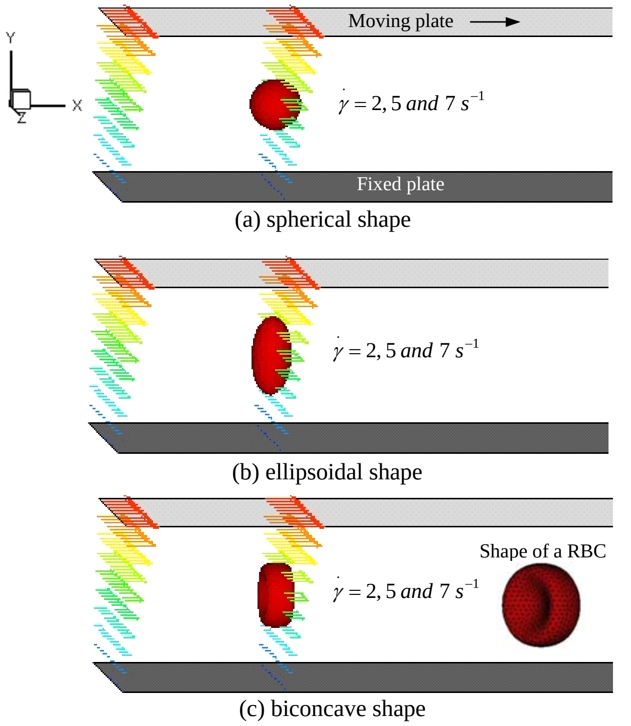


Fig. 1 Schematic of analysis models with spherical, ellipsoidal and biconcave shapes in a simple shear flow between two parallel plates for both transverse motion and rotation (MODEL 1)

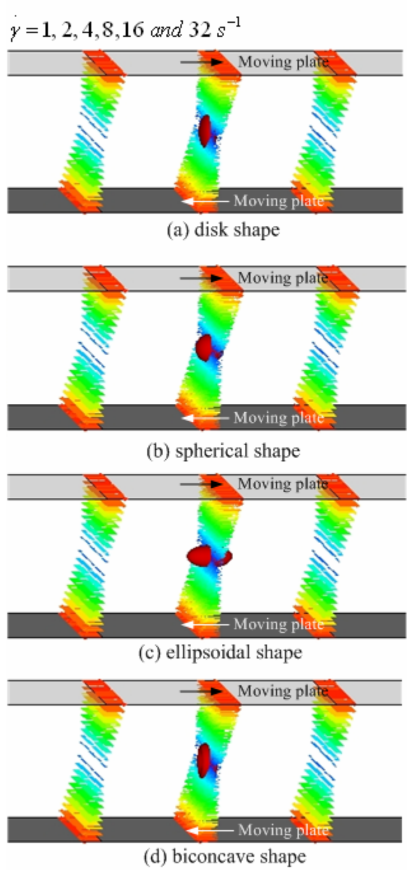


Fig. 2 Schematic of analysis models with shapes in a simple shear flow between two parallel plates for only rotation (MODEL 2)

adopting a control-volume-based technique (finite-volume method) to solve the Navier-Stokes equations (FLUENT).

An accurate solution of the Navier Stokes equations for deforming meshes is provided by the use of the ALE formulation, which makes it possible to include grid velocities in the momentum and continuity equation of the fluid domain. The ALE description conjugates Lagrangian and Eulerian features. The computational grid is neither moved with the boundary (Lagrangian) nor held fixed (Eulerian). Rather, it is moved in some arbitrarily specified way to give a continuous reconfiguration capability. Because of this freedom in moving the computational mesh offered by the ALE description, greater distortions of the continuum can be handled better than would be allowed by a purely Lagrangian method, with more resolution than that afforded by a purely Eulerian approach. The partitioned approach is used to simulate the interplay between plasma and RBCs. This strategy preserves the fluid and the structural solvers as separate. Both parts are alternately integrated in time and the interaction is taken into account by the boundary conditions of both the solvers. As a direct consequence there exists an intrinsic time lag between the integration of the fluid and the structure, which can be avoided by repeating the interaction until both the solution consistently produce the same result.

The general scheme of the coupling procedure is shown in Fig. 3. As mentioned above, the fluid domain is solved using the finite volume method computational code Fluent, which provides a number of features well suited to handle the specific problem of rotating boundaries. We will use a spring-based

moving, deforming mesh module, which allows a robust mesh deformation handling by assuming that the mesh element edges behave like an idealized network of interconnected springs. In order to maximize the influence of the boundary node displacements on the motion of the interior nodes, no damping was applied to the springs. To preserve the quality of the mesh during the valve motion, the maximum admissible skewness of the computational cells is set. The Fluent remeshing algorithm is adopted to properly treat degenerated cells, which agglomerates cells that violate the skewness criterion, and locally remeshes the agglomerated cells. If the new cells satisfy the skewness criterion, the mesh is locally updated with the new cells (with the solution interpolated from the old cells); otherwise, the new cells are discarded (FLUENT Users Manual, 2007).

The moving deforming mesh module is used in conjunction with two user-defined subroutines, named

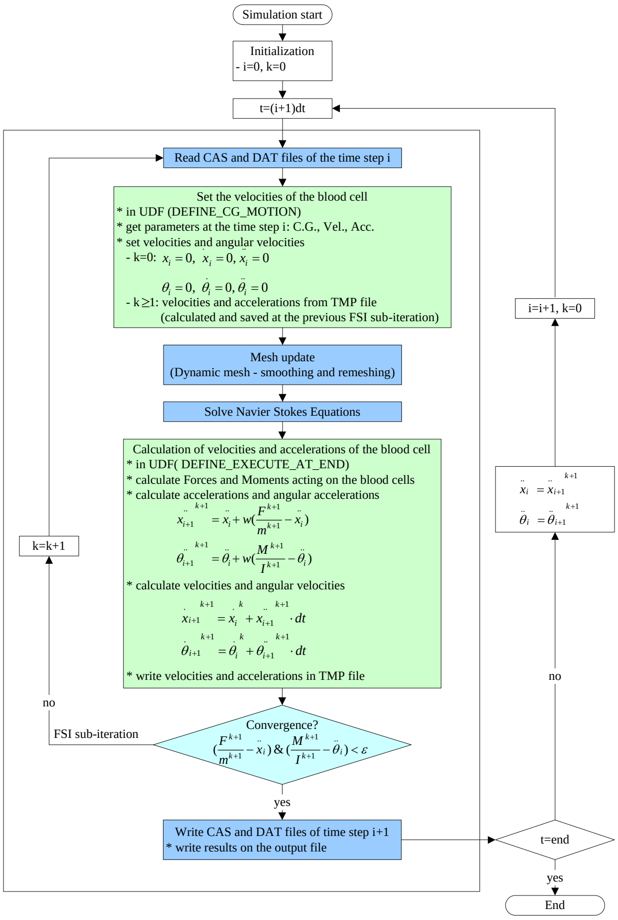


Fig. 3 Flow chart of the fluid-structure interaction algorithm using FLUENT (i: the time step number, k: the number of FSI iteration within i time step)

DEFINE_EXECUTE_AT_ END (moving body dynamics; MBD) and DEFINE_CG_MOTION (center of gravity motion; CGM), respectively; at the beginning of each step the first one calculates and updates the kinematics of the RBCs on the basis of the moment applied to the RBCs, which is calculated by the second subroutine at the end of the time step, once the time step convergence has been achieved. An iterative call to the fluid solver is performed by an external subroutine in order to update the solution of the fluid dynamic field and achieve the convergence of the FSI cycle, until the difference between the external momentum divided by the inertia of the fluid (calculated by MBD) and the angular acceleration (imposed by CGM) is not below a threshold value. More in detail, the transverse motion and rotation of RBCs are calculated with the following equations.

$$
\begin{aligned}
m\ddot{x} &= F_p + F_s \\
I\ddot{\theta} &= M_p + M_s
\end{aligned}
\tag{1}
$$

where m is the mass of the RBC, F_p and F_s are the pressure and shear forces acting on the RBC, respectively. I is the angular inertia, $\ddot{\theta}$ the angular acceleration, M_p the torque applied on the RBC by the pressure field, and M_s is the moment generated by shear stresses.

The acceleration value for the subsequent iteration within the generic time step i is updated through an under-relaxation scheme as

$$
\begin{aligned}
\ddot{x}_{i+1}^{k+1} &= \ddot{x}_i^{k} + w\left(\frac{F^{k+1}}{m^{k+1}} - \ddot{x}_i\right) \\
\ddot{\theta}_{i+1}^{k+1} &= \ddot{\theta}_i^{k} + w\left(\frac{M^{k+1}}{I^{k+1}} - \ddot{\theta}_i\right)
\end{aligned}
\tag{2}
$$

where k is the iteration index and w is the under-relaxation factor, which plays the role of damping changes in the acceleration produced during each iteration. Starting from the acceleration obtained in Eq. (2), the velocity and the displacement of the particle are calculated using the Newmark method.

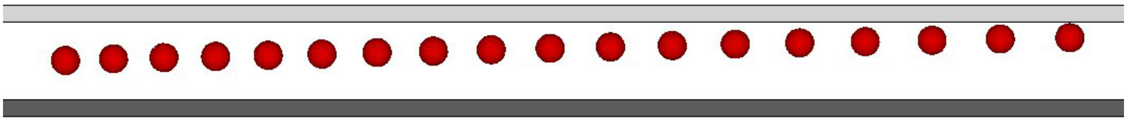
III. Results and Discussions

A. Model 1

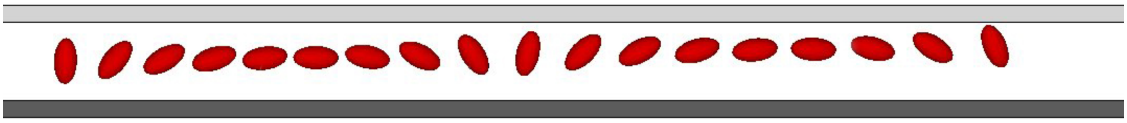
Series of the transverse motion and rotation of the particle with arbitrary shape suspending within the simple shear flow are presented in Fig. 4. Detailed information is shown in Fig. 5: displacements and velocities of the particles in the x-direction and the y-direction, and rotation angles and angular velocities. The results show that the shapes of the particles influence on the transverse motion and rotation of the particle suspending in simple shear flow.

B. Model 2

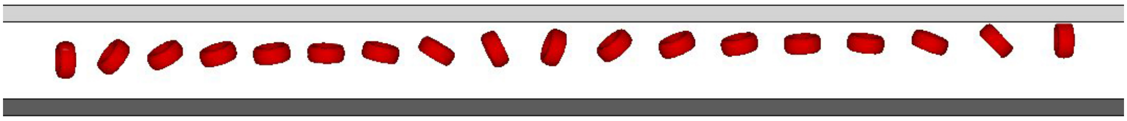
Velocity vector fields are presented in Fig. 6. Detailed information is shown in Fig. 7: rotation angles and angular velocities. The results show that the shapes of the particles influence on the rotation of the particle suspending in simple shear flow.



(a) spherical shape



(b) ellipsoidal shape



(c) biconcave shape

Fig. 4 Series of the transverse motion and rotation of the RBC suspending in the simple shear flow ($\dot{\gamma}$ =2 s^{-1}) (MODEL 1)

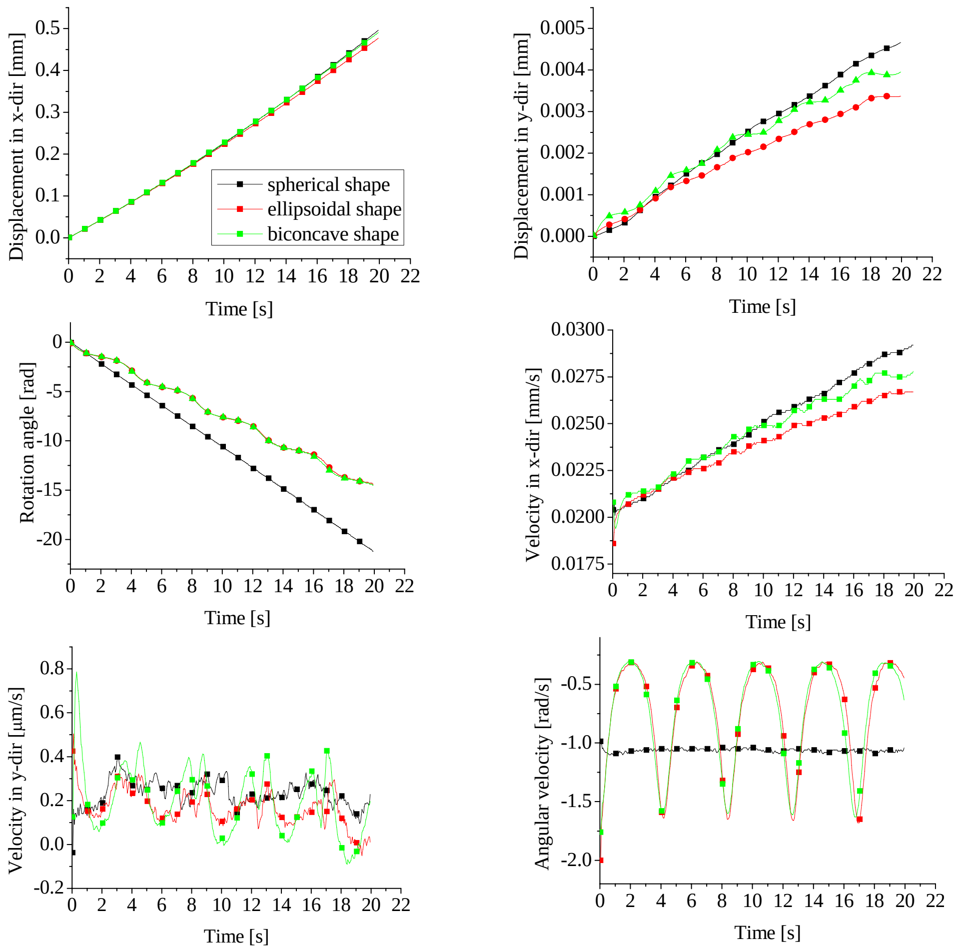
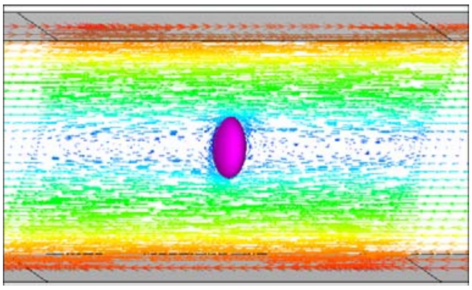
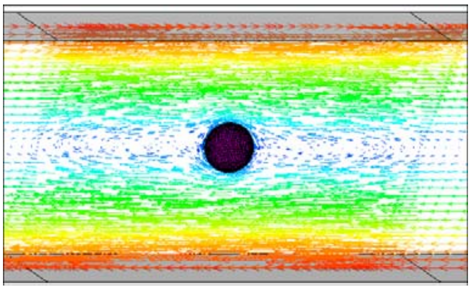


Fig. 5 The transverse motion and rotation of the RBC suspending in the simple shear flow ($\dot{\gamma}$ =2 s^{-1}) (MODEL 1)

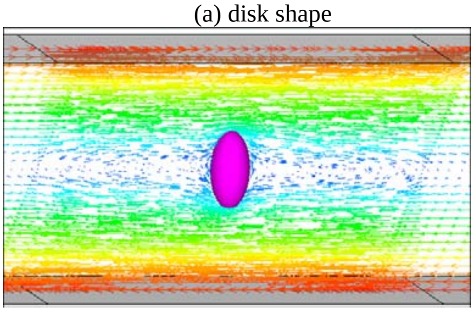
International Journal of Medical, Medicine and Health Sciences
ISSN: 2517-9969
Vol:3, No:1, 2009



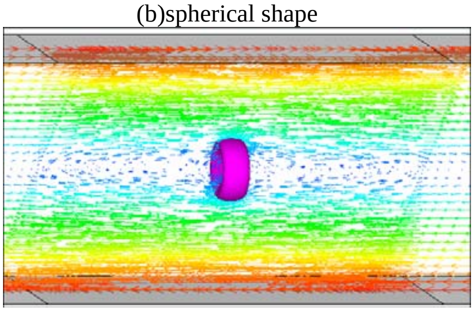
(a) disk shape



(b)spherical shape

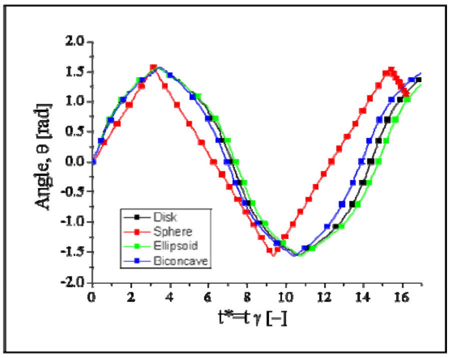


(c) ellipsoidal shape

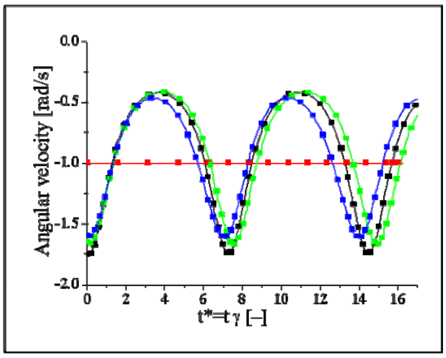


(d) biconcave shape

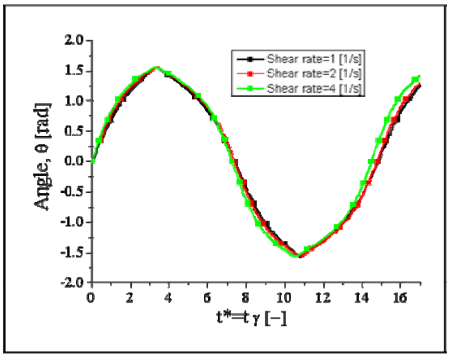
Fig. 6 Velocity vectors in the simple shear flow ($\dot{\gamma}$ =2 s^{-1}) (MODEL 2)



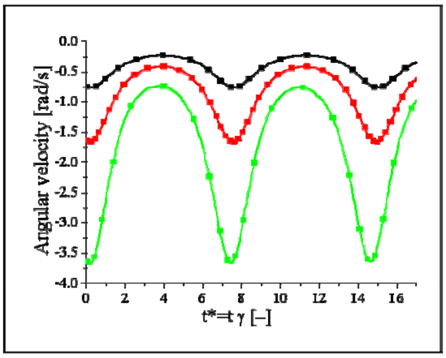
(a) angle ($\dot{\gamma}$ =2 s^{-1})



(b) angular velocity ($\dot{\gamma}$ =2 s^{-1})



(c) angle (ellipsoidal shape)



(d) angular velocity (ellipsoidal shape)

Fig. 7 The rotation of the RBC suspending in the simple shear flow (MODEL 2)

IV. Conclusion

In this study, the transverse motion and rotation of particles with arbitrary shape suspending within a simple shear flow have been successfully investigated using a fluid-structure interaction (FSI) method based on the Arbitrary-Lagrangian-Eulerian (ALE) approach and the dynamic mesh method (smoothing and remeshing) in FLUENT (ANSYS Inc., USA). The effects of the shapes on the transverse motion and rotation of the particles have been estimated.

The employed FSI method could be applied to the motions and deformations of a single blood cell and multiple blood cells, and the primary thrombogenesis caused by platelet aggregation. It is expected that, combined with a sophisticated large-scale computational technique, the simulation method will be useful for understanding the overall properties of blood flow from blood cellular level (microscopic) to the resulting rheological properties of blood as a mass (macroscopic).

References

[1] Goldsmith, H., Turitto, V., 1986. Rheological aspects of thrombosis and haemostasis: basic principles and applications. ICTH-Report-Subcommittee on Rheology of the International committee on thrombosis and haemostasis. Thrombosis and Haemostasis 55, 415-435.

[2] Miyazaki, H., Yamaguchi, T., 2003. Formation and destruction of primary thrombi under the influence of blood flow and von willebrand factor analysed by a D.E.M. Biorheology 40, 265-272.

[3] Wootton, D., Ku, D., 1999. Fluid mechanics of vascular systems, diseases, and thrombosis. Annual Review of Biomedical Engineering 1, 299-329.

[4] Chien, S., Usami, S., Skalak, R., 1984. Blood flow in small tubes. In: Handbook of Physiology-The Cardiovascular System IV, pp. 217-249.

[5] Shiga, T., Maeda, N., Kon, K., 1990. Erythrocyte rheology. Critical Review in Oncology/Hematology 10, 9-48.

[6] Pries, A., Neuhaus, D., Gaehtgens, P., 1992. Blood viscosity in tube flow: dependence on diameter and hematocrit. American Journal of Physiology 263, H1770-H1778.

[7] Mchedlishvili, G., Maeda, N., 2001. Blood flow structure related to red cell flow: a determination of blood fluidity in narrow microvessels. Japanese Journal of Physiology 51, 19-30.

[8] Botnar R, Rappitsch G, Scheidegger MB, Liepsch D, Perktold K, Boesiger P., 2000. Hemodynamics in the carotid artery bifurcation: a comparison between numerical simulations and in vitro measurements. J Biomech 33, 137-44.

[9] Hughes TH, Taylor C, Zarins C., 1998. Finite element modelling of blood flow in arteries. Comput Meth Appl Mech Eng 158, 155-96.

[10] Quarteroni A, Tuveri M, Veneziani A., 2000. Computational vascular fluid dynamics: problems, models and methods. Comput Visual Sci 2, 163-97.

[11] Sherwin SJ, Shah O, Doorly DJ, Peir_oJ, Papaharilaou Y, Watkins N, et al., 2000. The influence of out-of-plane geometry on the flow within a distal end-to-side anastomosis. ASME J Biomech 122, 1-10.