Blood Flow Simulation toward Actual Application at Hospital

R. Himeno [1]

1. *Advanced Center for Computing and Communication, RIKEN, Email: himeno@riken.jp*

Abstract

We started Computational Biomechanics Research project at RIKEN in 1999. In this project, we have developed several circulatory simulation codes, such as a fluid-structure coupling simulation code, blood flow simulation codes for the left ventricle and the aorta, kidney artery, carotid artery and stenosis in the brain. In addition, we have also developed both image processing code to handle medical images to get the shape of the artery and mesh generation code for simulation. Using these codes, we have become to estimate the effects of surgical operation such as putting the stent, removing stenosis and so on.

Keyword: *blood flow, fluid-structure coupling, medical imaging, mesh generation, Voxel*

1. Introduction

In many developed countries, diseases caused by blood flows such as heart attack, apoplectic stroke or cerebral infarction is one of the top death reasons. When a stenosis in coronary artery prevents the blood flow, it causes a heart attack. Once the surface of the stenosis is damaged, there should be blood clot, which may choke blood flow in cerebral artery or in coronary artery. In a varix, the blood may be stagnant, which makes blood clotted. Then the clot may be transferred to lung and choke the blood tube there. Therefore it is very important for us to know how blood is flowing in those areas. To know how blood flows will help doctors or surgeons to make a diagnosis or to make an operation plan. When we started the Computational Biomechanics Research Project at RIKEN in 1999, we started developing not only flow solvers but also a shape acquisition system. In this paper, those flow solvers and some of the computed results as well as the shape acquisition system will be shown.

2. Non-Newtonian Influence on Blood Flows in Artery

The blood flow is well known non-Newtonian flow. However, this non-Newtonian effect is strong only in capillary. Blood flows in artery have small influence of the non-Newtonian effect. To see this first, we simulated flows in an artery with a stenosis with and without non-Newtonian effect.

2.1 Computational Method and Meshes

Fluid is supposed as incompressible and governed by the equation of continuity and the Navier-Stokes equations whose viscosity may not be constant.

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)\, u = -\frac{1}{\rho} \triangle p + \nu \triangle^2 u \tag{2}$$

The MAC method [1] is used to solve these equations. The equations are transfered into curvilinear coordinate system and discritized by finite difference method. The third-ordered upwind-difference method is used for the convective terms and second ordered difference method is used for other spatial differencing terms. The first-ordered Euler implicit method is used for the time-differencing term [2, 3].

A straight circular pipe with hemisphere bump is used as a model of the artery with stenosis here. Let the diameter of the tube as D, the length of the pipe is 14 D and the diameter of the hemisphere is D.

The height of the hemisphere is 0.5 D as shown in Fig.1. The total number of the meshes is 117,760 (115 x 32 x 32).

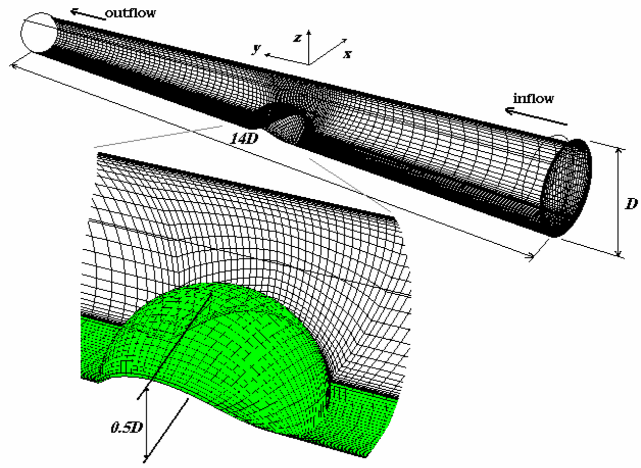

Fig.1 Boundary fitted grid system for the flow in an artery with a non-axisymmetric stenosis.

2.2 Non-Newtonian Model for Blood Flow

The non-Newtonian effect is said to be weak in arteries but it is not obvious that the effect near the stenosis is also weak. To see the difference, we calculated flows with constant viscosity and blood viscosity model. Ishikawa proposed a blood viscosity model based on Bingham fluid but it estimates too much viscosity in low deformation speed. Watanabe, who was in our research project, proposed a model which is improved this point and is used here to see the difference between ordinary Newtonian fluid and the blood [4].

$$\mu_k = \mu_k(e_{ij}) = \mu_0 + \frac{\mu_y - \mu_0}{\exp\left(\beta\left|e_{ij}\right|\right)} \tag{3}$$

Even absolute value of the shear rate tensor e_{ij} becomes zero in this model, the viscosity stays μ_y. Fig. 2 shows this model compare with other two blood models.

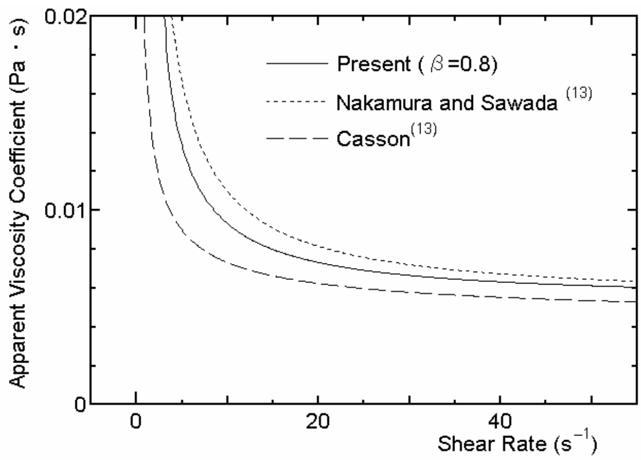

Fig. 2 Apparent viscosity coefficient variation of the present constitutive equation.

2.3 Non-Newtonian Effects in Steady Flow

Fig. 3 shows streamlines and surface pressure field of time averaged flow field in the artery with the stenosis at Reynolds number 1,000. In this case, the difference between Newtonian and non-Newtonian fluids is very small and hard to see. However, to see the surface shear stress on the stenosis, there is about 10 % difference in the peak value shown in Fig. 4.

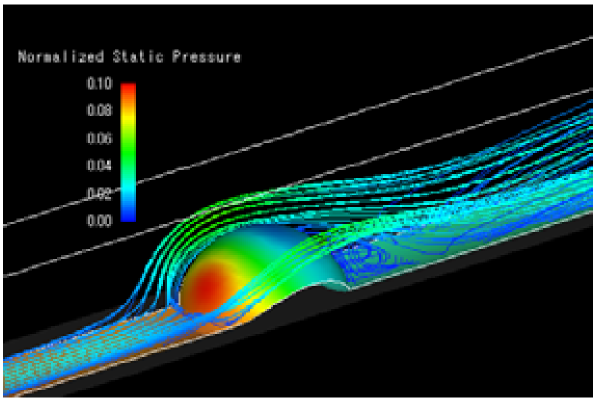

Fig. 3 Numerical result for a steady inflow case, (Reynolds number: 1000, normalized pressure distribution and stream lines).

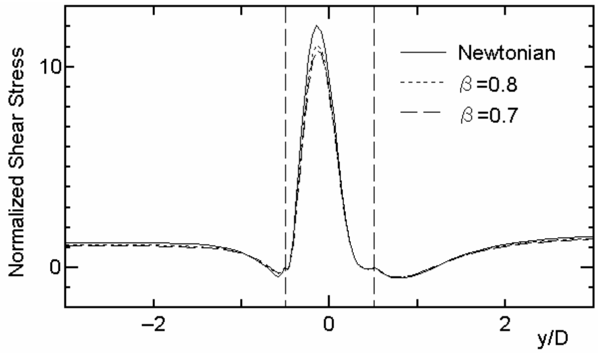

Fig. 4 Normalized wall shear stress distributions on a center line of the stenosis (steady inflow cases)

2.4 Non-Newtonian Effects in Pulsatile Flows

The results of the steady flow case show the non-Newtonian effect is small except for the peak shear stress. In this section, the results of the pulsatile case is presented and discussed. Fig. 5 shows the inflow condition of the calculation. Reynolds number is 1,000 and Womersley number is 3.9 in this case. To see these results, it is found that the Newtonian effect in the artery is small and negligible.

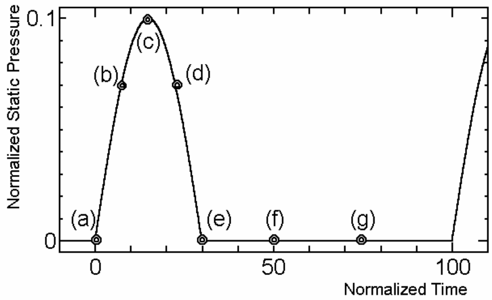

Fig. 5 Pulsatile inflow condition (Womersley number : 3.9)

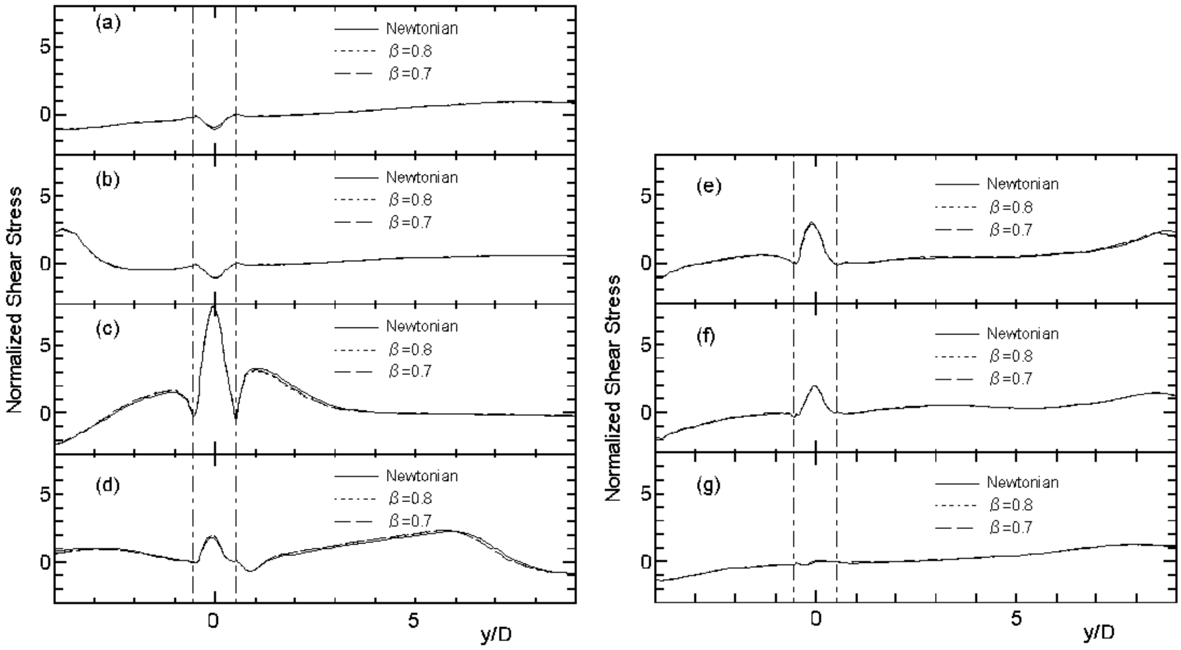

Fig. 6 Surface shear stress at the center line of the artery with the stenosis.

3 Fluid-Structure Coupled Simulation

In the case of the pulsatile flows in our body, the artery expands its diameter when the heart beats. It is necessary to simulate this dynamic change of the artery caused by the flow. To simulate it, weak coupling approach is used here. At first, the boundary shape is supposed as fixed and the flow is solved by the flow solver. Then we know the force distribution acting on the boundary surface and simulate the displacement of the boundary by a structure analysis solver. These processes are continued to get periodic solutions. Fig. 7 shows the shape change and surface pressure of the artery with the stenosis. In this case, boundary displacement is calculated by using Maxell model. This calculation is a preliminary results to check the argolithm. The displacement shown here is too much because Poisson rate is 10 times small of the blood tube's property.

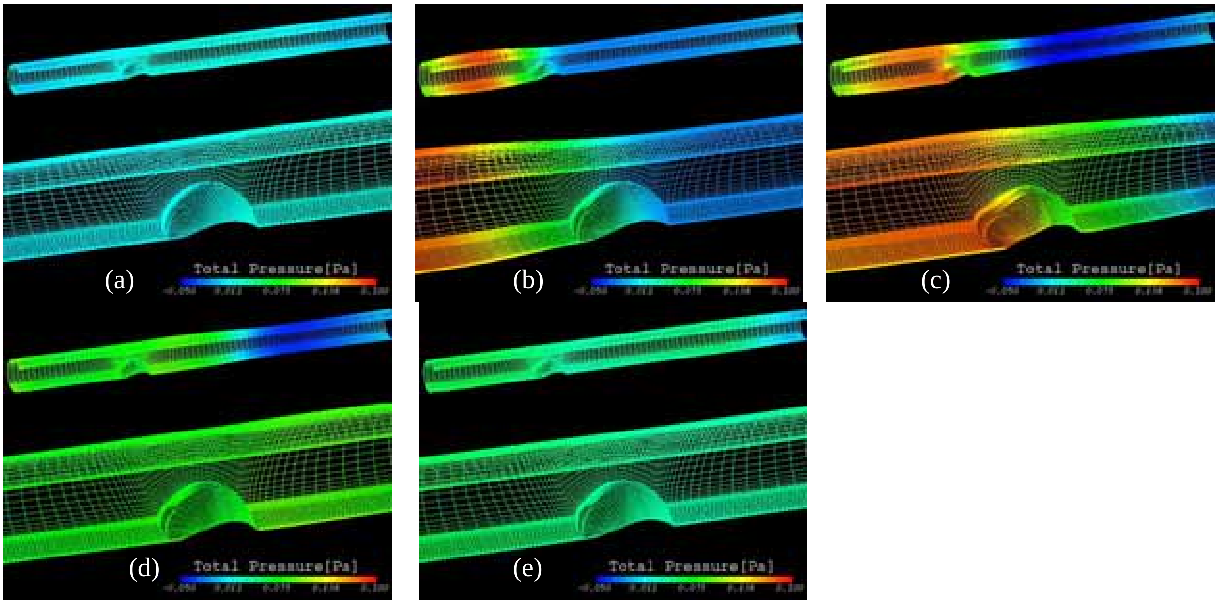

Fig. 7 Surface pressure and deformation of blood tube and stenosis..

4 Beating Heart and Flow Simulation

There are several works to simulate beating heart. Recently, very fast CT system which can get series of the heart shape has been developed. But, such kind of CT system is too expensive for ordinary hospitals and not popular. They usually use Ultrasonic Imaging system which is not expensive. We have developed a blood flow analysis system for the left ventricle based on a series of extracted shape of the left ventricle. Fig. 8 shows the process of getting moving meshes from the medical images and fig.9 shows the flow filed in the left ventricle. In this model, velocity boundary conditions at the inlet and outlet were also given by a ultrasonic velocimeter.

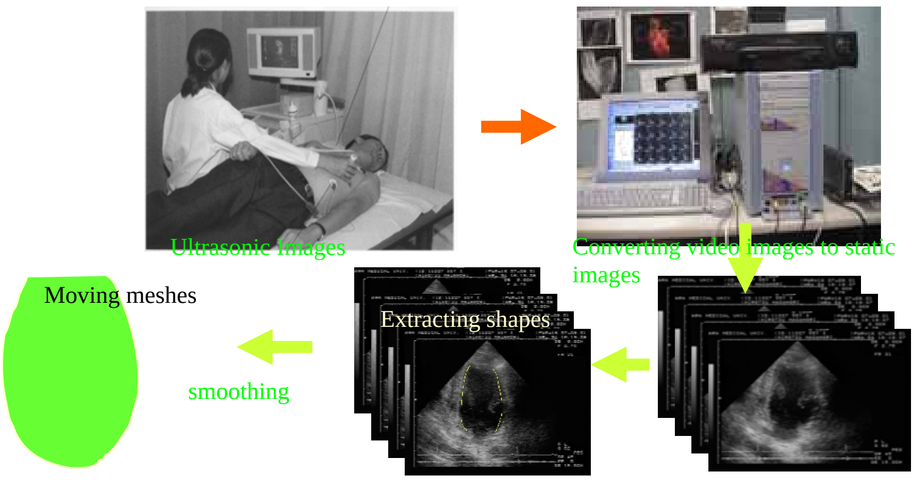

Fig.8 Process of getting moving mesh of left ventricle from the medical images

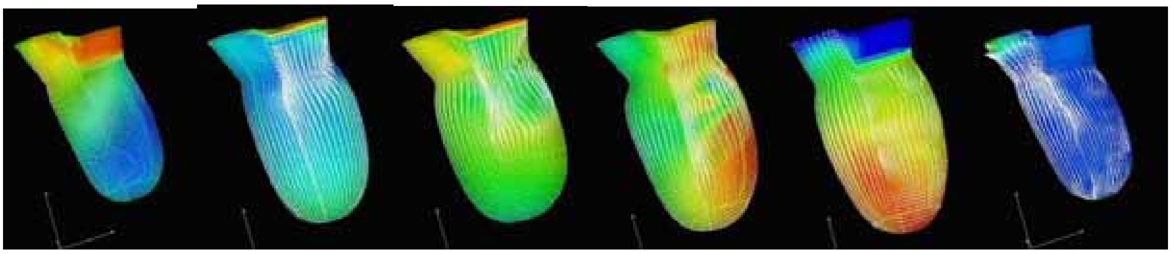

Fig. 9 Flow vectors and pressure distribution in the left ventricle

5 Other Artery Cases

Overlapping grid technique is used for simulating aorta with multi-branches shown in Fig.10. The boundary conditions are given by one-dimensional simulation. Fig. 11 (a) and (b) shows flow fields in a carotid artery with stenosis. The simulated results show this stenosis makes surface shear stress 30% larger than that of no stenosis case.

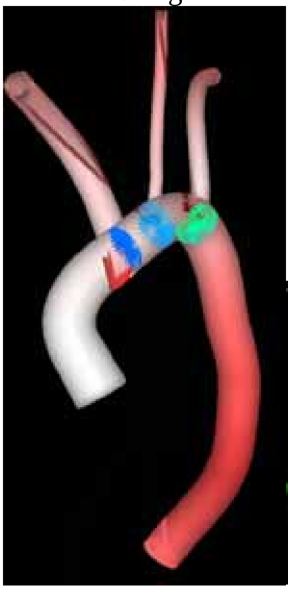

Fig.10 Flow in aorta

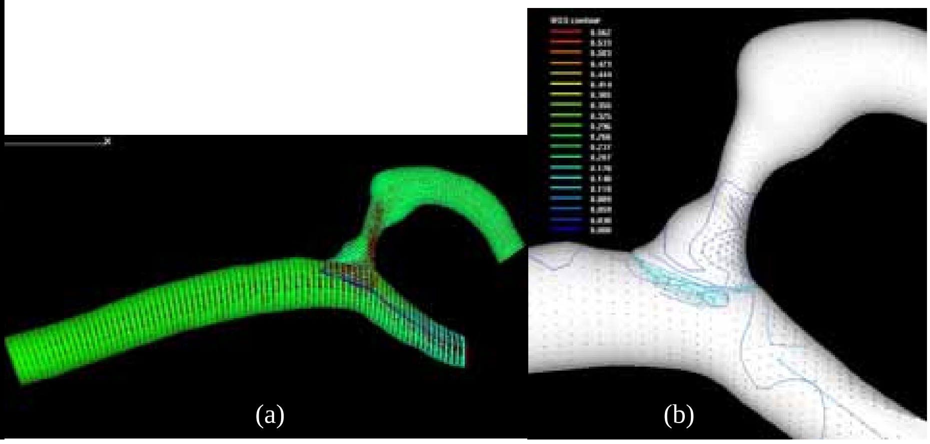

Fig. 11 Flow fields in carotid artery with a stenosis

6 Simulation of Catheter Operation for Cerebral Artery Aneurysm

When an aneurysm is found in cerebral artery and it needs treatment, a coil or stent is used for this operation. Fig. 12 and 13 show simulation of these cases. Fig. 14 shows a catheter simulator for training.

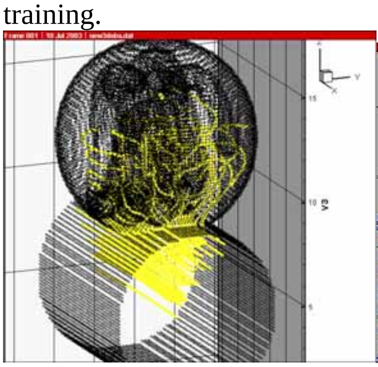

Fig.12 Flow around the coil.

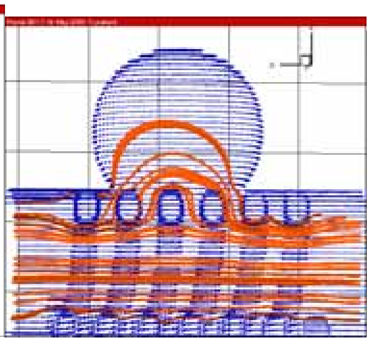

Fig.13 Flow through stent.

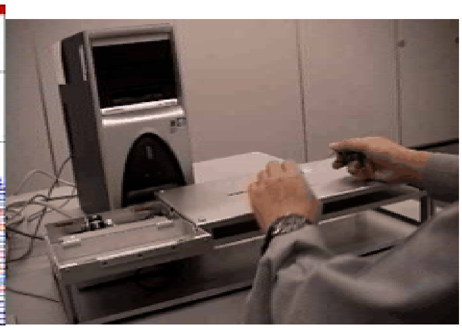

Fig.14 Catheter simulator

7 Conclusion

We have developed several simulation system for investigation of hemo-dynamics in human circulatory system, which include medical image processing system and several operation tools used in hospitals. We hope the system will become widely used in hospitals in near future.

Acknowledgement

The author would like to express my sincere gratitude to not only all project members in the Computational Biomechanics Project at Riken but also researchers in universities and doctors in hospitals.

References

[1] Roache, P. J., "Computational Fluid Dynamics", Hermora Publishers, 1976.

[2] Himeno, R., Takagi, M., Fujitani, K. and Tanaka, H., "*Numerical Analysis of the Airflow around Automobiles Using Multi-block Structured Grids*", SAE, SAE Technical Paper Series 900319 (1990) .

[3] Himeno, R., "*Implementation of an Incompressible Navier-Stokes Solver on Vector / Parallel Computers and Application to the Aerodynamic Analysis of Automobiles*", Automotive Applications of Vector / Parallel Computers : State of the Art, M. Ginsberg (editor), SP-923, SAE Inc. U.S.A., pp.1-6（1992）

[4] Watanabe, N. and Himeno, R., "A Three-Dimensional Numerical Analysis for a Pulsetile Flow in a Non-Aximmetric Stenosed Tube", Proceedings of the 13th CFD Symposium, Tokyo (1992)

[5] Proceddings of Riken Symposium, "Computational Biomechanics", May 2000.

[6] Proceddings of Riken Symposium, "Computational Biomechanics", Jun 2001.

[7] Proceddings of Riken Symposium, "Computational Biomechanics", July 2002.

[8] Proceddings of Riken Symposium, "Computational Biomechanics", May 2003.