11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

Verification of Porous Loss Model for Assessment of Flow Diverters

Takashi Suzuki[1], Syo Kadokura[1], Hiroyuki Takao[2], Satoshi Tateshima[3], Shunsuke Masuda[2], Dahmani Chihebeddine[2,4], Yi Qian[5], Fernando Vinuela[3], Yuichi Murayama[2,3] and Makoto Yamamoto[6]

[1] Graduate School of Mechanical Engineering, Tokyo University of Science
6-3-1 Niijyuku, Katsushika-ku, Tokyo, 125-8585, Japan, j4512636@ed.tus.ac.jp
[2] Department of Neurosurgery, Jikei University School of Medicine
3-25-8 Nishishinbashi, Minato-ku, Tokyo, 105-8461, Japan, takao@jikei.ac.jp
[3] Division of Interventional Neuroradiology, David Geffen School of Medicine at University of California Los Angeles, 10833 Le Conte Avenue, Los Angeles, CA 90095, USA
[4] Siemens Japan K.K.
Gate City Osaki West Tower, 11-1 Osaki 1-Chome, Shinagawa-ku, Tokyo, 141-8644, Japan
[5] The Australian School of Advanced Medicine, Macquarie University
NSW 2109, Australia
[6] Department of Mechanical Engineering, Tokyo University of Science
6-3-1 Niijyuku, Katsushika-ku, Tokyo, 125-8585, Japan, yamamoto@rs.kagu.tus.ac.jp

Key Words: *Cerebral Aneurysm, Flow Diverter, CFD, Porous Loss Model*

Recently flow diverters (below FDs) have been developed for the treatment of cerebral aneurysms. Operators deploy these devices in the parent blood vessel to divert the blood flow away from the aneurysm itself. However non-occluded and ruptured cases after treatment indicate the existence of unresolved problems. To investigate the effects of FDs, Computational Fluid Dynamics (CFD) have been performed. To simulate the blood flow using FD geometries, it needs higher machine specifications because of the larger number of computational elements due to the large scale difference between the size of the flow diverter's struts and the aneurysms. For the alternative computational method, the modeling of FDs as a porous loss model was proposed[1]. As a first step of the assessment of FDs using CFD, we also validate the applicability of the porous model for different kinds of FDs.

We performed patient-specific CFD simulations using 3D models obtained from digital subtraction angiography. Two patients treated with one of Pipeline Embolization Device (PED; ev3/Covidien) were analyzed. The blood flow was simulated using ANSYS CFX 14.5 (ANSYS). We compared two methods for reproducing the FD in computations as follows:

1) Geometric model

PED geometries fitted within the parent vessels were created using Amira (Visualization Sciences Group). And these geometries were imported as a stationary immersed solid in the simulation.

2) Directional loss model like a porous region[1]

We modeled PED as a porous region by the addition of a momentum source term to the N-S equations. A streamwise-oriented coordinate system (x',y',z') such that the x' axis is aligned with the streamwise direction and y', z' axes lie on the transverse planewas used. The directional loss model is describes as follows:

$$S_{M,x'} = -\frac{\mu}{K^S_{perm}} U_{x'} - K^S_{loss} \frac{\rho}{2} |\boldsymbol{U}| U_{x'} \quad (1)$$

$$S_{M,y'} = -\frac{\mu}{K^T_{perm}} U_{y'} - K^T_{loss} \frac{\rho}{2} |\boldsymbol{U}| U_{y'} \quad (2)$$

$$S_{M,z'} = -\frac{\mu}{K^T_{perm}} U_{z'} - K^T_{loss} \frac{\rho}{2} |\boldsymbol{U}| U_{z'} \quad (3)$$

where, K^S_{perm}=1.9735×10^{-10} m^2 and K^T_{perm} =2.5232×10^{-10} m^{-1} are the streamwise and transverse permeabilities, and K^S_{loss}=2.9622×10^4 m^{-1} and K^T_{loss} are the streamwise and transverse loss coefficients. Each coefficients were obtained via numerical simulations[1]. Concerning the transverse losses, we assumed that viscous losses were dominant, therefore inertial losses were neglected. In the flow field we assumed laminar flow since the Reynolds number based on the blood vessels diameter was around 500. The mass flow rate waveform[2] was imposed at the inlet boundary. At the outflow boundary, pressure was fixed to 0 Pa. Rigid and non-slip boundary conditions were assumed on all the vascular walls. The blood was assumed to be a Newtonian fluid with density and viscosity of 1,100 kg/m^3 and 0.0036 Pa·s respectively. The unsteady flow analysis were performed over two heartbeats (1.8 sec) with a time step of 5×10^{-4} sec. The data of last one cycle were investigated in detail.

Figure 1 compares the streamline visualizations at peak systole from two methods. Table 1 summarizes the values of the volume-average velocity in the aneurysm and the area-average pressure at the aneurysm surface. These values are the average of one cardiac cycle. It indicates that the result using the porous model overestimates diversion effects qualitatively and quantitatively. The pressure are in good agreement.

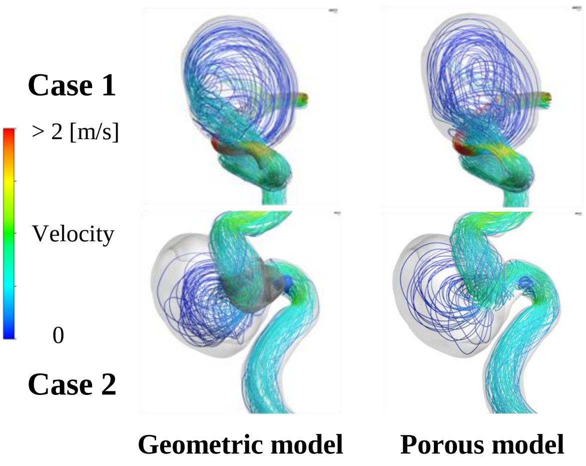

Fig. 1 Streamline visualization

Table 1 Time averaged parameters

		Geometric model	Porous model	relative error [%]
case 1	Velocity inside the IA [m/s]	0.0659	0.0439	32.2
case 2		0.0283	0.0145	48.8
case 1	Pressure at the IA surface [Pa]	3091	3216	4.1
case 2		3118	3136	0.6

*IA: Intracranial Aneurysm

REFERENCES

[1] L. Augsburger, P. Reymond, D. A. Rufenacht and N. Stergiopulos, Intracranial Stents Being Modeled as a Porous Medium: Flow Simulation in Stented Cerebral Aneurysms. *Ann Biomed Eng.*, Vol. **39**(2), pp. 850–863, 2010.

[2] M. D. Ford, N. Alperin, S. H. Lee, D. W. Holdsworth, and D. A. Steinman, Characterization of volumetric flow rate waveforms in the normal internal carotid and vertebral arteries. Physiol. Meas., Vol. **26**, pp. 477-488, 2005.