심혈관 혈류역학 시뮬레이션을 위한 임상적으로 실용적인 CFD 해석 툴 개발

웬민투안[*] · 최형권[**] · 이상욱[***]

From bench to bedside: The development of a clinically practical CFD tool for cardiovascular flow dynamics

Minh Tuan Nguyen[*], Hyoung G. Choi[**], Sang-Wook Lee[***]

Abstract : We introduce the development of clinically practical CFD tool for complex cardiovascular flow dynamics simulations with aims of accelerating clinical outcomes, basically by bench to bedside approach. Fully implicit P2-P1 finite element code was developed using four-step fractional scheme with stabilization technique of SUPG and parallel computing by domain decomposition method. The techniques enables patient-specific hemodynamic solution to be available within a half hour's computing time in most of vascular flow cases.

1. Introduction

It is widely accepted that local hemodynamics plays an important role in initiation and progression of cardiovascular diseases such as atherosclerosis and aneurysm(1). While the computational fluid dynamics (CFD) technique has emerged in the past decade as a promising tool for studying, in a subject-specific manner, detailed cardiovascular flow dynamics, it still has some barrier to directly enter the clinical uses in the diagnosis and treatment planning procedure due to its impractical computing time and numerical instability. However, advancement of numerical algorithms in CFD and high-performance parallel computing enables to translate this powerful engineering tool to regular clinical process.

In this paper, we introduce the development of clinically practical CFD tool for cardiovascular flow dynamics simulations with aims of accelerating clinical outcomes, basically by bench to bedside approach.

*   울산대학교 일반대학원 기계공학과
**   서울과학기술대학교 기계자동차공학과
*** 울산대학교 기계공학부

2. Numerical Methods

2.1 Numerical Algorithm

Fully implicit P2-P1 finite element code was developed using four-step fractional scheme introduced by Choi and Moin(2).

step 1: $$\alpha^2 \frac{\hat{u}_i - u_i^n}{\Delta t} + Re\,\frac{1}{2}\left(\hat{u}_j \hat{u}_{i,j} + u_j^n u_{i,j}^n\right) = -Re\,p_{,i}^n + \frac{1}{2}\left(\hat{u}_{i,jj} + u_{i,jj}^n\right)$$

step 2: $$\alpha^2 \frac{u_i^* - \hat{u}_i}{\Delta t} = Re\,p_{,i}^n$$

step 3: $$p_{,ii}^{n+1} = \frac{\alpha^2}{Re\Delta t} u_{i,i}^*$$

step 4: $$\alpha^2 \frac{u_i^{n+1} - u_i^*}{\Delta t} = -Re\,p_{,i}^{n+1}$$

In this scheme, all terms in step 1 are discretized in time using Crank-Nicolson method which treats the convection and diffusion terms implicitly. In order to prevent numerical instabilities in solving high Reynolds number flow, the streamline-upwind/ Petrov-Galerkin formulation was applied.

2.2 MPI Parallel Algorithm

The computational domain is decomposed into multiple subdomains and the computing work for each subdomain is distributed to the corresponding processor and runs simultaneously. Fig. 1 illustrates the subdomain matrix structure using the domain decomposition method, where M_i is mass matrix, N is convection matrix, L_i and L_i^T are gradient and divergence matrix respectively, and C is Laplacian matrix (i represents x, y, z direction).

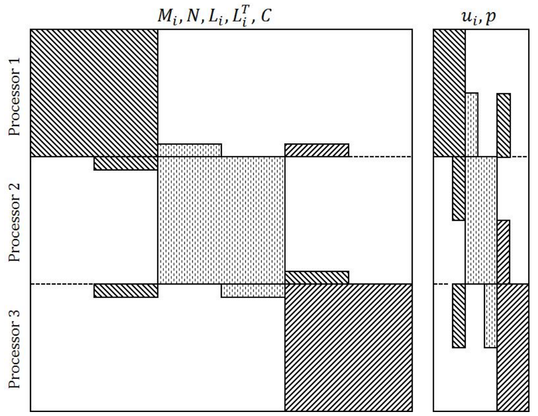

Fig. 1 Subdomain non-zero pattern of matrix and variables for four-step fractional method

2.3 Integrated Multiscale Modeling (3D+0D)

The accuracy of 3D CFD solutions based on patient-specific data highly depends on the boundary conditions, not only at the inlet, but also, at outlets. However, in vivo data from each patient are not always available. To estimate physiologically adequate boundary conditions, lumped parameter models (LPM, 0D models) were adapted for all outlets and aorta inlet and then 3D CFD modeling for local hemodynamics and 0D modeling for the rest of distal vascular system were implicitly coupled. Fig. 2 shows the typical LPM for coronary vascular systems. The procedure to determine the value of resistance and capacitance in coronary vascular system can be found in detail in the study of Sankaran[3].

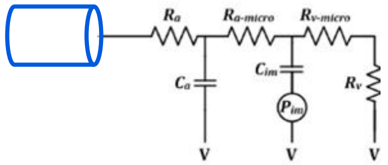

Fig. 2 Lumped parameter model of coronary vascular systems

4. Conclusion

We developed a fast, robust and physiologically reasonable multiscale CFD code for complex blood flow dynamics simulations, which may be useful in routine clinical practices. In order to improve the accuracy, LPM modeling techniques for each outlet branch based on patient-specific data need to be studied further.

Acknowledgement

This work was supported by IITP grant (No. R0101-15-0171) funded by the Korea government (MSIP)

Reference

(1) Malek, A. M. et al., 1999, "Hemodynamic shear stress and its role in atherosclerosis," JAMA, Vol. 282, pp. 2035-2042

(2) Choi, H. et al., 1994, “Effects of the computational time step on numerical solutions of turbulent flow,” Journal of Computational Physics. Vol. 113, pp. 1-4.

(3) Sankaran et al., 2012, “Patient-specific multi-scale modeling of blood flow for coronary artery bypass graft surgery,” Annals of Biomedical Engineering, Vol. 40, pp. 2228-2242.