PATIENT SPECIFIC PARALLEL CFD ANALYSIS FOR CARDIOVASCULAR SYSTEM SIMULATIONS

Anastasios Skiadopoulos[1], Evangelos Makris[1,2], Panagiotis Neofytou[1], Sokrates Tsangaris[2], Christos Housiadas[1]

[1] Thermal Hydraulics & Multiphase Flow Laboratory, NCSR "Demokritos", Agia Paraskevi, Greece; [2] Laboratory of Biofluid Mechanics & Biomedical Engineering, School of Mech. Engineering, NTUA, Zografou, Greece

Introduction

Patient specific computational simulation of blood flow in the cardiovascular system can be a very powerful tool in the field of medical diagnosis and prognosis. In this study, a reliable and time-efficient computational fluid dynamics (CFD) methodology that provides accurate blood flow simulations in the cardiovascular system is presented.

Methods

The proposed methodology comprises of the following steps. Initially, the patient specific geometry is obtained using an in-house code, which processes medical imaging data and generates multi-block structured computational grids from medical images [Makris, 2011].

Simulations are, then, carried out by a parallelized in-house CFD code employing the Semi- Implicit Method for Pressure Linked Equations (SIMPLE) algorithm in conjunction with a collocated Finite Volume Method (FVM) in Cartesian coordinates [Neofytou, 2006]. This code is suited for multi-block and non-orthogonal computational grids and incorporates various non- Newtonian models simulating the material behaviour of blood [Neofytou, 2008].

Parallelization of the algorithm is achieved via a domain decomposition technique, applying a Single Program Multiple Data (SPMD) style of parallel implementation. Communication between processes is accomplished through the Message Passing Interface (MPI) message passing standard.

Results

The proposed methodology is illustrated through the simulation of blood flow in a patient specific arterial segment. The flow is regarded as pulsating, laminar and incompressible, whereas blood is assumed a Newtonian fluid. The algorithm is able to simulate accurately the pressure and velocity field. The significant speedup achieved by the algorithm parallelization renders the method very useful for medical prognosis and diagnosis.

Discussion

A patient specific parallel CFD methodology for the cardiovascular system is developed in order to provide a useful computational tool for medical assessment on realistic clinical time scales. As a test case, the blood flow in an arterial segment obtained by medical imaging (patient specific approach) is simulated.

Implementation of hybrid parallelization schemes and fully integrating and interconnecting the various software components is the subject of future work, in order to further speedup the algorithm and study more complicated elements of the cardiovascular system.

References

Makris *et al*, Comput Method Biomech Biomed Eng, 15(2): 173-83, 2012.

Neofytou *et al*, Int J Numer Method Fluids, 51:489-510, 2006.

Neofytou *et al*, Comput Method Biomech Biomed Eng, 11:615-626, 2008.