TOWARDS THE USE OF LARGE-EDDY SIMULATIONS FOR THE PREDICTION OF THE BLOOD FLOW IN ARTIFICIAL ORGANS

Simon Mendez$^1$, Christophe Chnafa$^1$ and Franck Nicoud$^1$

$^1$ I3M, University Montpellier II, Place Eugène Bataillon, 34095 Montpellier, France
smendez@um2.fr - http://www.math.univ-montp2.fr/~mendez
christophe.chnafa@univ-montp2.fr - http://www.math.univ-montp2.fr/~chnafa
franck.nicoud@univ-montp2.fr - http://www.math.univ-montp2.fr/~nicoud

Key words: Large Eddy Simulations, Blood Flows, Artificial Organs, Immersed Boundary Method, Residence Time.

With the joint progresses of computing resources, medical imaging and numerical methods, cardiovascular computational fluid dynamics (CFD) has been dealing with more and more complex configurations with an increasing accuracy. Among the most important directions for research, one can cite the understanding of cardiovascular flows in healthy and diseased conditions, CFD-aided patient-specific surgical planning, enhanced diagnosis and design/optimization of biomedical devices. Design and optimization of medical devices is an extremely challenging field of research, due to the complexity of the (moving) geometrical features, the nature of the flow, sometimes transitional or even turbulent and the multi-scale character of the problem. Both macroscopic and cellular constraints have to be dealt with to optimize devices in which blood flows. Indeed, the use of biomedical devices is plagued by thrombosis and hemolysis, which are small-scale phenomena. Applying CFD in this context is so complicated that the standard tool to study biomedical devices remains the laboratory experiment [2].

In the present study, we present a numerical framework appropriate for accounting for the transitional nature of blood flows in large vessels, organs or biomedical devices. The YALES2BIO solver (http://www.math.univ-montp2.fr/~yales2bio/) used here is a fully explicit, massively parallel, 3D CFD in-house code dedicated to the computation of microscopic and macroscopic cardiovascular flows [1, 4]. It relies on the YALES2 solver (http://www.coria-cfd.fr/index.php/YALES2), widely validated for complex engineering applications [5]. It is based on low-dissipative, fourth-order finite-volume approximations and an explicit fourth-order Runge-Kutta scheme for time integration. Because of the moderate Reynolds number and the possible transitional nature of the flow, the Large Eddy Simulation (LES) approach is followed. An advanced subgrid scale model able to represent the proper turbulence damping near solid walls is used [6]. YALES2BIO
includes an Arbitrary Lagrangian-Eulerian method and an unstructured immersed boundary method to account for deformable domains.

The first objective of the paper is to stress the relevance of the LES approach in the context of ‘high’ Reynolds number blood flows, with the application to artificial organs. Figure 1 shows a volume-rendering of the vorticity in a patient-specific left heart [1], performed with YALES2BIO. This calculation will be used to demonstrate the importance of cycle-to-cycle variations on the assessment of hemolytic and thrombotic risks. The second objective is to show the versatility of the YALES2BIO solver by discussing results from simulations of biomedical devices such as cytometers using the Coulter principle [3] flow diverters used in the treatment of aneurysms.

References