Computers & Fluids 94 (2014) 173–187

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Image-based large-eddy simulation in a realistic left heart

C. Chnafa *, S. Mendez, F. Nicoud

University Montpellier II, I3M, CNRS UMR 5149, Place Eugene Bataillon, 34095 Montpellier cedex 5, France

ARTICLE INFO

Article history:
Received 11 July 2013
Received in revised form 17 December 2013
Accepted 27 January 2014
Available online 18 February 2014

Keywords:
Patient-specific
Hemodynamics
Heart
Turbulence
Image registration
Arbitrary Lagrangian–Eulerian

ABSTRACT

A numerical framework allowing insight in fluid dynamics inside patient-specific human hearts is presented. The heart cavities and their wall dynamics are extracted from medical images, with the help of a non-linear image registration algorithm, in order to obtain a patient-specific moving numerical domain. Flow equations are written on a conformal moving computational domain, using an Arbitrary Lagrangian–Eulerian framework. Resulting equations are solved numerically with a fourth-order finite-volume technique. Application of this framework to compute a patient-specific left heart flow is presented as well. The blood flow is characterized by its transitional nature, resulting in a complex cyclic flow. Flow dynamics is analysed in order to reveal the main fluid phenomena and to obtain insights into the physiological patterns commonly detected. It is demonstrated that the flow is neither laminar nor fully turbulent, thus justifying a posteriori the use of Large Eddy Simulation.

1. Introduction

Intracardiac hemodynamics is closely related to the morphology and function of the heart: changes in the heart shape or in its wall kinetics alter the blood flow patterns. Therefore, analyzing the blood flow spatial and temporal distribution in the heart may provide information on cardiac abnormalities. However, in clinical routine, hemodynamics is mostly observed indirectly through global variables as the cardiac output in order to assess the cardiac performance. Indeed, a synthetic description of the available information and its relation with the heart function is still lacking. Note also that the hemodynamics analysis may not only improve early diagnosis but also open up new perspectives for the understanding of cardiovascular physiology.

Recent technological innovations in imaging techniques have provided valuable opportunities for non-invasive assessment of hemodynamics. Blood flow velocities can be measured in vivo using phase-contrast magnetic resonance imaging (PC-MRI) or by echocardiography techniques.

PC-MRI studies have contributed to the understanding of hemodynamic features [1–7]. Although very comprehensive, the PC-MRI velocity mapping is not real-time. Hence, beat-to-beat variations in the flow cannot be recorded (the *k*-space is filled over many cardiac cycles). Moreover, PC-MRI suffers from a relatively low spatio-temporal resolution, precluding the observation of small-scale and fast time-varying flow features [8,9].

Echocardiography techniques [10,11], with higher spatio-temporal resolution make an alternative to PC-MRI. However, echocardiography only gives access to velocity components directed towards or away from the ultrasonic beam, while one would want to measure the full 3D flow vectors. Nevertheless, investigations have been conducted on normal and abnormal hearts leading to potential hemodynamics-based biomarkers for cardiac health. [12–14].

In order to obtain more information about the heart hemodynamics, in vitro investigations have been performed in fully controlled experiments [15–18]. Blood patterns in heart chambers replications have been studied thanks to particle image velocimetry in healthy and abnormal configurations.

In addition to these studies, computational fluid dynamics (CFD) has been more and more used to predict blood flow in the heart over the last decade. In silico replications of heart chambers, mainly the left ventricle (LV), have been considered. Simulations in idealized ventricles [19–21] or in more realistic geometry [22] have been performed. As in vitro experiments, such fundamental CFD studies are particularly useful to isolate and elucidate the effect of well-controlled parameters on the blood flow. However, inherent simplifications raise the question of the relevance of their conclusions for individual clinical cases. In this context, numerical simulation using a combination of computational methods and medical imaging techniques for determining vascular geometry

Abbreviations: PC-MRI, phase-contrast magnetic resonance imaging; CFD, computational fluid dynamics; LV, left ventricle; EFSI, electrical–fluid–structure interaction; CT, Computed Tomography; LA, left atrium; AO, Aorta; RSPV, right superior pulmonary vein; RIPV, right inferior pulmonary vein; LSPV, left superior pulmonary vein; LIPV, left inferior pulmonary vein; MV, mitral valve; AV, aortic valve; NS, Navier–Stokes; FKE, fluctuating kinetic energy.

* Corresponding author. Tel.: +33 4 67 14 92 97.
E-mail address: Christophe.Chnafa@univ-montp2.fr (C. Chnafa).

http://dx.doi.org/10.1016/j.compfluid.2014.01.030
0045-7930/

appears to be a relevant strategy. CFD starts to be a mature technique for arterial flows [23–26], but its application to heart haemodynamics faces additional challenges:

- the geometry of the blood domain is complex and it undergoes large deformations,
- opening and closing valves make the topology of the domain change over the cardiac cycle,
- the flow is the result of a complex electrical–fluid–structure interaction problem, and
- the flow regime is most probably transitional between laminar and turbulent and varies over the cardiac cycle.

Two main different strategies have been developed to obtain simulation of the blood flow in realistic heart geometries. The most natural one is to extract the heart geometry at one chosen moment in the heart cycle and to solve an electrical–fluid–structure interaction (EFSI) problem [27–32]. In this approach, patient-specific data are needed [33,34]. What is the exact rheology of the myocardial muscle? What is the load produced by the heart environment? How to reproduce the mechano-electric coupling in the heart muscle? All these questions make such an approach extremely challenging. Another strategy consists in using realistic heart wall movements extracted from cine MRI or Computed Tomography (CT) scan data. Heart movement is not computed, but prescribed from the patient-specific medical images, which can be acquired using standard clinical imaging procedures. Such a computational approach, where the geometry and the movements are extracted from images will be referred to as image-based computational fluid dynamic (IB-CFD). Different research teams have developed IB-CFD methods for heart flows, more specifically to study the left ventricle alone [35–38]. Recently, more advanced work has been published, using a full heart model obtained from CT images [39] or a heart model fed from MR images [40]. The feasibility of cardiac IB-CFD has been shown, but the flow results notably suffered from limited spatial resolution or partial geometries (LV only in the majority of the cases). Furthermore, there has been remarkably little focus on the presence of turbulence in the heart, except in experimental works [16,18].

In the present paper, an image-based CFD method is presented. As in the aforementioned IB-CFD works [35–38], medical images are used to generate a moving patient-specific domain, in which the blood flow equations are solved. The geometry movements are generated from a 4D sequence (MRI or CT scan images) treated by an appropriate image registration algorithm [41,42]. This approach has been used before to compute blood flow in aortas [43]. It is further developed for application to the left heart flow, notably by introducing valve modelling. In order to demonstrate the ability of the method to compute flow in the heart, its application to a full patient left heart using 4D CT scan images is presented. Extensive description of the flow obtained is provided and the main flow characteristics usually reported in the literature are emphasized. First numerical insight into turbulence in the left heart is reported as well.

The numerical method is detailed in Section 2. The characteristics of the applied case is presented in Section 3 and the flow field obtained is described in Section 4. Concluding remarks are given in Section 5.

2. Methodology

In this section, the image-based computational fluid dynamics approach is detailed. First, the fluid problem resolution is detailed in an ALE framework. Next, Section 2.2 presents how the moving computational domain is obtained from the medical images. Specific valve modelling is needed and is the object of Section 2.3. Finally the method to obtain the inflow and outflow boundary conditions is described.

2.1. Fluid problem

2.1.1. Governing equations

Blood can be modelled as an incompressible fluid, but red blood cells induce a complex rheological behaviour [44]. However, for high stress levels and in large vessels, non-Newtonian effects are usually neglected and blood is usually modelled as an incompressible Newtonian fluid in numerical simulations [24,39]. Taking the incompressible flow assumption into account and assuming blood as a Newtonian fluid, the fluid motion is described by the Navier–Stokes (NS) equations. These equations are solved on the moving blood domain $\Omega_f(t) \subset \mathbf{R}^3$ bounded by $\partial\Omega_f(t)$. The boundary $\partial\Omega_f(t)$ is such that $\partial\Omega_f(t) = \partial\Omega_f^i(t) \cup \partial\Omega_f^w(t) \cup \partial\Omega_f^o(t)$ and $\partial\Omega_f^i(t) \cap \partial\Omega_f^w(t) \cap \partial\Omega_f^o(t) = \emptyset$ where $\partial\Omega_f^i(t)$ represents a fluid inlet boundary where a Dirichlet condition is prescribed on the velocity field, $\partial\Omega_f^w(t)$ represents the vessels and heart wall boundary and $\partial\Omega_f^o(t)$ represents a fluid outlet boundary. The NS equations read:

$$\left.\begin{aligned} \tfrac{\partial \mathbf{u}_f}{\partial t} + (\mathbf{u}_f \cdot \nabla)\mathbf{u}_f &= -\tfrac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u}_f + \mathbf{f}, \\ \nabla \cdot \mathbf{u}_f &= 0, \end{aligned}\right\} \quad \text{on} \quad \Omega_f(t) \tag{1}$$

where $\mathbf{u}_f$ is the fluid velocity field, p is the pressure field, ν the kinematic viscosity, ρ the density and $\mathbf{f}$ a volumetric force. The corresponding initial and boundary conditions are,

$$\mathbf{u}_f(\mathbf{x}, 0) = \mathbf{u}_f^0(\mathbf{x}) \quad \text{on} \quad \Omega_f(0), \tag{2}$$

$$\mathbf{u}_f(\mathbf{x}, t)|_{\mathbf{x}\in\partial\Omega_f^w(t)} = \mathbf{u}_s(\mathbf{x}, t) \quad \text{on} \quad \partial\Omega_f^w(t), \tag{3}$$

$$\mathbf{u}_f(\mathbf{x}, t)|_{\mathbf{x}\in\partial\Omega_f^i(t)} = -U^{in}(\mathbf{x}, t)\mathbf{n_o}(\mathbf{x}) \quad \text{on} \quad \partial\Omega_f^i(t), \tag{4}$$

where $U^{in}(\mathbf{x}, t)$ is the inlet velocity profile imposed as a Dirichlet condition, $\mathbf{n_o}$ the outward normal at the inlet faces and $\mathbf{u_s}$ is the endocardium surface velocity field imposed as a Dirichlet condition as well. A convective outlet boundary condition is imposed on $\partial\Omega_f^o(t)$ as,

$$\frac{\partial \mathbf{u}_f(\mathbf{x}, t)}{\partial t} + U^{conv}\frac{\partial \mathbf{u}_f(\mathbf{x}, t)}{\partial \mathbf{n}} = \mathbf{0}, \tag{5}$$

where $\mathbf{n}$ is the outward normal at the outlet patch and U^{conv} the convective velocity. The uniform convective velocity U^{conv} is imposed in such a way to meet the global mass conservation over $\Omega_f(t)$. The surface velocity $\mathbf{u_s}$ is not computed but extracted from the medical images and applied as boundary conditions for the fluid problem (see Section 2.2.2).

2.1.2. Time advancement scheme

The time advancement scheme is an explicit low-storage four-step Runge–Kutta scheme [45] recast in an ALE formalism and coupled with the Chorin's projection correction method [46] for the pressure term. The grid is displaced during the prediction step only.

The starting point for deriving the time-advancement scheme is the integral form of the NS equations on a node-centred control volume $\omega(t)$, its boundary S_t moving with the mesh velocity $\mathbf{u}_g$. The equations integrated in time between $t^n = t^0 + n\Delta t$ (Δt being the time step size) and t_{n+1} are classically [47]:

$$\int_{t^n}^{t^{n+1}} \frac{\mathrm{d}}{\mathrm{d}t}\int_{\omega(t)} \mathbf{u}\ \mathrm{d}\omega\ \mathrm{d}t + \int_{t^n}^{t^{n+1}}\int_{\omega(t)} \nabla\cdot((\mathbf{u}_f - \mathbf{u}_g)\mathbf{u}_f)\ \mathrm{d}\omega\ \mathrm{d}t = \mathbf{RHS}, \tag{6}$$

where the **RHS** containing the viscous fluxes and the pressure gradient is omitted in the following. The four sub-steps of the time advancement are computed as: