Fluid-Structure-Interaction in a Beating Human Heart Model

A. Aksenov[1], V.Pokhilko[1], A.Yushenko[1], B.Butz[2], P.Sridhar[2], K.D'Souza[2], W.Zietak[3]

[1]TESIS (CAPVIDIA), Moscow, Russia

[2]Dassault Systemes Simulia Corporation, Johnston, RI, USA

[3]CAPVIDIA, Belgium

1   Abstract

Cardiovascular disease is the leading cause of death in developed countries and continues to drive significant research aimed at improving diagnosis and treatment. In cases where therapeutic medical device intervention is warranted, it is important to account for device-heart interaction over both the short and long term. In the short term, cardiac motion can significantly impact device stability and performance (as in the case of artificial heart valves), while over the long term, modified blood flow patterns may lead to structural remodeling that can have adverse effects on cardiovascular function. Computational tools used for device design, sizing, and placement must therefore be able to account for cardiac/vascular tissue mechanics, blood flow, and the interaction between them. Moreover, such tools must allow for patient-specific modeling and provide interactive visualization capabilities that can support clinical decision-making and reduce operator error.

In this paper, we describe a methodology for bidirectional fluid-structure interaction between the general purpose CFD code *FlowVision* and the SIMULIA *Living Heart Human Model (LHHM)*, a dynamic, anatomically realistic, four-chamber heart model that considers the interplay and coupling of electrical and mechanical fields acting in concert to regulate the heart filling, ejection, and overall pump functions. The LHHM natively includes a 1D fluid network model capable of representing dynamic pressure/volume changes in the intra- and extra-cardiac circulation. In the current work, we first replace this network model with a full 3D blood model (solved in FlowVision) that provides detailed spatial and temporal resolution of cardiac hemodynamics driven by motions of the beating heart and constrained with appropriate time-varying boundary conditions derived from the literature. After validating this approach, we activate bidirectional coupling between the blood flow CFD model and the LHHM electromechanical model using the SIMULIA co-simulation engine and discuss modeling details and results of interest.

2   Introduction

The application of numerical simulation methods to model and simulate human heart flow makes it feasible to design and test medical devices in a timely and low cost manner. Moreover, we can deeply analyze the heart function under different (abnormal) conditions. While the LHHM has demonstrated the feasibility of modeling the structural behavior of the heart, fluid-structure interaction (FSI) of this system presents several challenges. From a standalone CFD perspective, key challenges include (1) meshing and performance issues arising from the complex geometry of the heart; (2) specifying accurate boundary conditions (to represent the extracardiac circulation) that are location-dependent, time-varying, and adaptive; and (3) handling the large deformations of the mesh as the heart contracts and relaxes over each heartbeat. FSI introduces further challenges such as (1) dynamic mesh mapping between the nonconforming fluid and structural meshes; (2) accurate modeling of the wetted interface; and (3) overcoming computational issues such as robustness and performance. Nonetheless, in recent years, the application of CFD to cardiac hemodynamics has made great progress and is even being applied in clinical decision making. While many models exist that can adequately predict flow behavior in specific regions of the cardiovascular system and under specific conditions, they are often based on research codes and/or require expert knowhow and parameter tuning. The intent of this work was to use commercially available models and tools and to systematically develop a methodology to enable non-expert users in the medical device and clinical community to better understand the cardiac system and explore treatment options. This paper discusses the initial efforts towards this goal - a methodology for bidirectional FSI between the general purpose CFD code FlowVision and the SIMULIA Living Heart Human Model (LHHM).

3 Methods

3.1 Previous work: CFD Analysis using Moving Wall Method

To overcome the aforementioned challenges of modeling CFD in the heart, a simulation with moving wall boundaries was performed. The simulation covered a full cardiac cycle and was based on 100 CT scan images from a real patient (Fig.1). The time-dependent geometry changes were applied as moving wall boundaries conditions derived from the Wiggers diagram (Fig. 4).
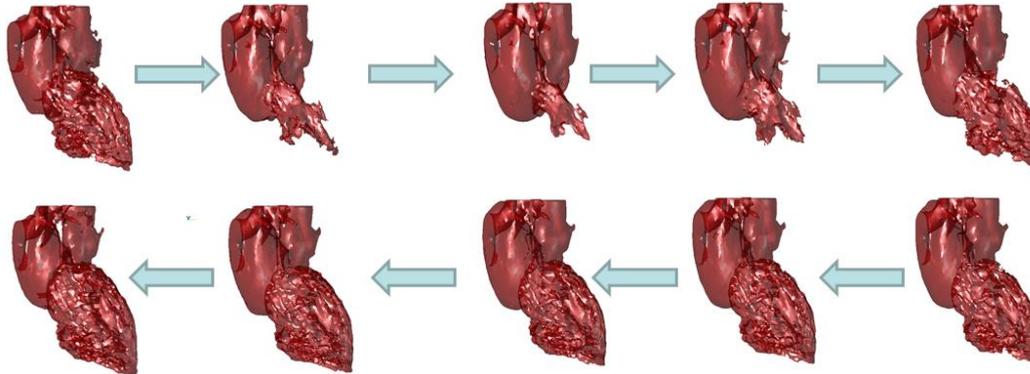

Fig. 1: Cardiac motion over a single heartbeat: selected frames from real patient CT scan data

After having successfully demonstrated the feasibility of the simulation in FlowVision (Fig. 2), the next step was to couple the flow solver to the electromechanical LHHM.
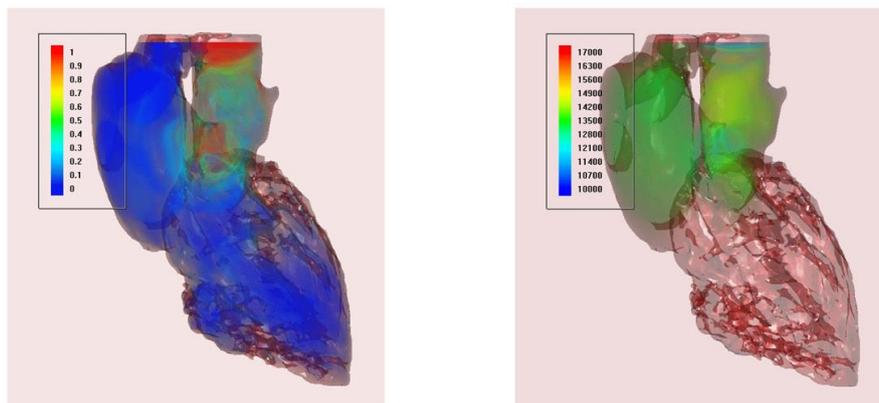

Fig. 2: Blood flow velocity (left) and pressure (right) within the heart derived from patient CT scan data

3.2 Living Heart Human Model (LHHM)

The LHHM is a high fidelity representation of a normal (healthy) 4-chamber adult male human heart. It includes well defined anatomic details of the atria, ventricles, heart valves and proximal vasculature, such as the aortic arch, pulmonary artery, and superior vena cava (Fig.3). The dynamic response of the Heart Model is governed by a realistic representation of the electrical and mechanical physics. An initial implicit analysis computes the electrical potential distribution in space and time, and a subsequent explicit analysis computes the mechanical displacement using the electric potentials as the excitation sources. In the basic (native) model, blood flow is modeled by a system of fluid chambers and fluid links (representing heart valves and extracardiac circulation) and driven by volume changes computed in the mechanical analysis. This lumped parameter approach is computationally efficient and able to capture average dynamic quantities such as chamber pressures and volumes; as such, it has been successfully applied in several applications where *in vivo* mechanical effects are dominant, such as extending the fatigue life of pacemaker leads and coronary stents. However, in some applications, it is also necessary to understand *in vivo* localized hemodynamic quantities such as flow velocity and wall shear stress. To examine these effects, a full 3D CFD solver is required – hence the next step was to replace the network blood flow model with a full CFD model in FlowVision.
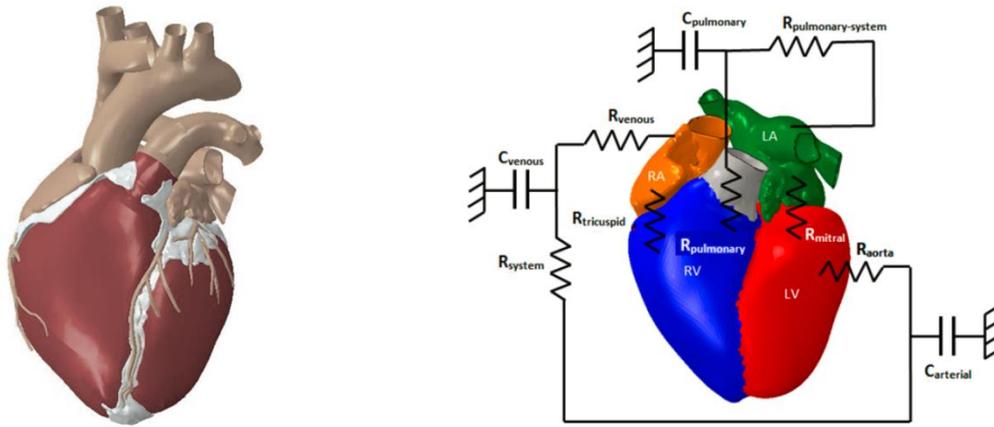
Fig. 3: LHHM - external geometry (left) and graphical representation of the blood flow model (right)

3.3 CFD model of the heart

The numerical simulation of the blood flow through the heart is computed by solving the full Navier-Stokes continuity equations for incompressible fluid, which are integrated in the FlowVision software. FlowVision uses a fully automatic mesh generator based on sub-grid geometry resolution method (SGGR) (Aksenov, 1998) [1]. With the heart shape changing in heartbeat process, it will also reconstruct the mesh dynamically in real-time. Thus the total number of cells is dynamic and ranges from 2.0 to 3.2 million polyhedral elements.

While blood is a non-Newtonian fluid, it is reasonable to model it as Newtonian within vessels of diameter larger than 1 mm (Noutchie, 2005) [2], which is the case in human heart chambers. Considering the low speed of the internal flow, it is also reasonable to model the flow as incompressible and without heat transfer. Taking into consideration the size of the heart chambers and the blood velocity, the standard k-ε turbulence model was used (Rubenstein, 2015) [3]. A time step size of 0.001s was selected to achieve adequate accuracy and reasonable performance. Time-dependent boundary conditions were defined for inlet and outlet – as in the native LHHM, these conditions were derived from the Wiggers Diagram (Figure 4). The outer boundaries of the model were defined as no-slip walls.
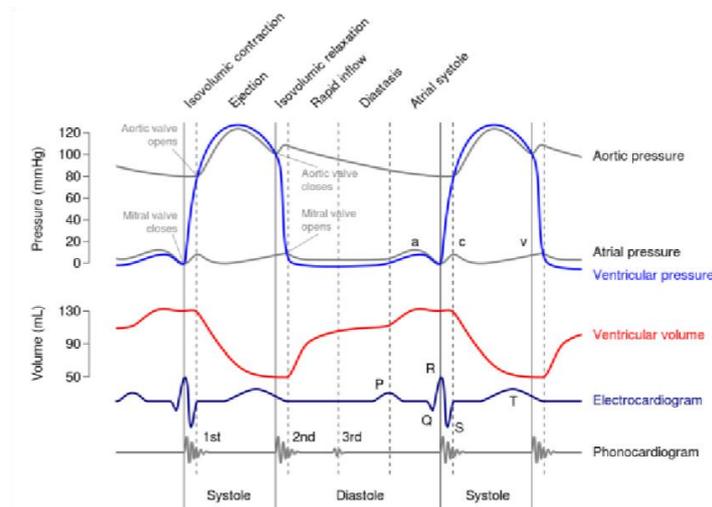

Fig. 4: Wiggers Diagram used to derive boundary conditions for the CFD model

3.4 Fluid-Structure Interaction (FSI)

FSI simulation between Abaqus and FlowVision was performed using the SIMULIA Co-Simulation Engine (CSE). After a test run of the Abaqus model, the wetted surface was extracted, inspected and modified to eliminate gap. It was then used to generate outer boundary of the CFD model. The FSI interface was generated using FlowVision mapper. During the co-simulation run, fluid and structural model exchange data at the FSI interface uses an explicit coupling scheme.

4   Results

Cardiac hemodynamics is driven both by the motion of the myocardium as well as by the motions of the four heart valves. To simplify the problem, the initial FSI simulations considered only the myocardial motion while neglecting the internal valves. The resulting CFD mesh deformations at select times within the cardiac cycle are shown in Fig. 5. Fig. 6 shows the internal flow patterns at 3 time points. In general, the flow field is unsteady and shows significant vorticity. Blood flow velocity reaches the maximum approximately at t=0.2s, which corresponds to the peak pressure. In a cardiac cycle, blood flow velocity is extremely slow, ranging from 0 to 1 m/s. During systole, the average velocity is greater than in diastole period. The peak velocity at different stages matches the results published in the reference literature (Wexler, 1968) [4].
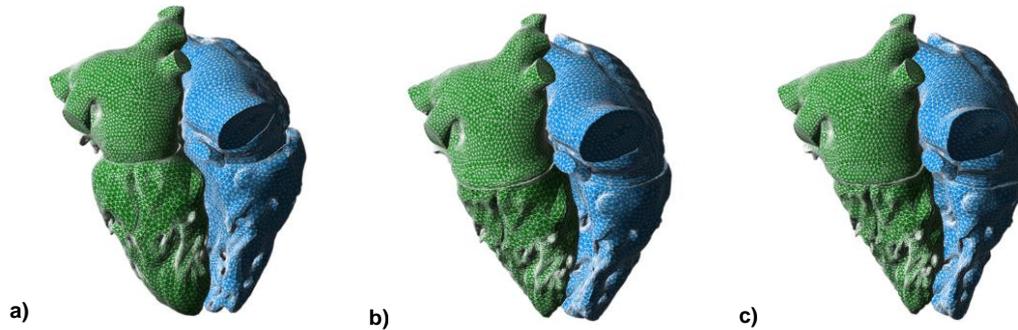

a)   b)   c)

Fig. 5: CFD mesh deformations at different time points of the FSI simulation a) 0.2s, b) 0.6 s, c) 0.8s
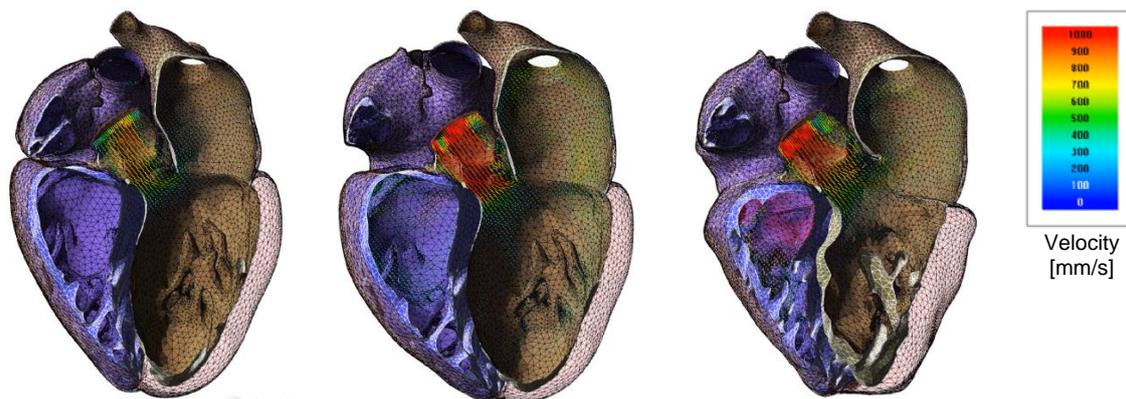

Velocity [mm/s]

Fig. 6: Blood flow velocity at different time points in the FSI simulation

5   Conclusions and Next Steps

In this paper, a novel approach for simulating the cardiac hemodynamics using the LHHM and FlowVision was presented. Having demonstrated the viability of full FSI of the heart, the next steps are to include the heart valves as they exert a strong influence on intracardiac flow velocities and pressures. While the overall results from the current model show reasonable comparison with published data, we expect that including the valves will not only improve the accuracy but also provide valuable insights with practical benefits for the device development and clinical community.

6   References

[1] Aksenov A, Dyadkin A, Pokhilko V. Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method // Proc. 1998 ASME Pressure Vessels and Piping Division Conference, San Diego, ASME PVP-Vol. 377-2., 1998. pp 79-86.
[2] S. C. O. Noutchie, "Flow of a Newtonian fluid: The case of blood in large arteries," Master of Science, University of South Africa, November 2005.
[3] David Rubenstein, Wei Yin, Mary D. Frame, "Biofluid Mechanics: An Introduction to Fluid Mechanics", London: Elsevier, second edition 2015, pp. 466
[4] Wilcox D. C. (1994) "Turbulence modeling for CFD", DCW Industries, Inc., 460 p.