Simulating blood flow in Living Heart Model

A. Aksenov[1)], L. Begisheva[1)], R. Cotton[2)], T. Luniewski[1)], D. Vucinic[3)], W. Zietak[1)]

[1)]CAPVIDIA, Belgium [2)] Simpleware, UK [3)]Vrije Universiteit Brussel, Belgium

Abstract*: Being able to visualize blood flow in leaving heart helps to understand heart diseases and define strategy for surgical intervention. The ideal simulation should be performed on a specific patient heart model (individual geometry) to correctly asses the specific cardiac problem and predict possible treatment. In this paper two approaches is presented for simulating blood flow using numerical CFD approach: simulation based on the SIMULIA Living Heart FE Model and simulation based on dynamic 3D heart model obtained from a 3D MRI/MRT scanner. The CFD simulation is performed in both cases using FlowVision CFD code. Both approaches provide a better understanding of the blood flow during a cardiac cycle. The simulation using dynamic heart model from 3D MRI has the advantage to be patient specific and can be used for diagnoses and definition of treatment strategy for a specific patient. The simulation based on the Living Heart FE Model is more suitable for function and performance evaluation of medical devices such as stunts or artificial heart values to understand and predict their influence on blood circulation system. Both approaches open new possibilities for cardiac related applications. As both methods are in a very early stage of the development it is difficult to cover and predict all potential applications areas and benefits at this stage. This paper is intended to start the dialogue between cardiologists and engineers to challenge the limits of current technology*.

Keywords*:Biomedics, two-way CFD Coupling, Living Heart Model, Computational Fluid Dynamics, Fluid-Structure Interaction, blood flow*.

1. Introduction

Today, CVDs (Cardio Vascular Disorder) are the first cause of death. By 2030, CVD-related deaths will reach 23.3 million each year (Noutchie , 2005). The lack of realistic human models limits ability to predict device behavior under realistic use conditions. Ultimately, 95% of all new devices are not tested in a human environment before approval. Growth in recalls suggests room for improvement.

Through numerical simulation method to modeling the human heart flow makes it feasible to design or test an artificial medical device more efficient and low-cost. Moreover, we can deeply study the heart function under different even abnormal conditions, and it will be more convenient to visualize the internal flow field of heart.

One heart cycle contains two phases: diastole (ventricle filling) and systole (ventricle contraction). Atrium filled with fresh blood. During diastole, the atria and ventricles of the heart relax and begin to fill with blood. The atria then begin to relax. Next, the heart's ventricles contract (ventricular systole) and pump blood out of your heart. At the end of diastole, the heart's atria contract (atrial systole) and pump blood into the ventricles. A heartbeat may seem like a simple, repeated event. However, it's a complex series of very precise and coordinated events. These events take place inside and around the heart.

A typical heartbeat cycle can be illustrated in Fig. 1, it can be noticed that the shape and volume always keep changing during the process. This brings some challenges for CFD simulation, which are difficult to be solved with traditional numerical method, and they are (1) the time-variant shape of geometry having big deformations requiring high-quality mesh deformation detection for its remeshing; (2) the boundary conditions are changing in time, and from Wiggers Diagram (a standard diagram used in cardiac physiology) we can extract the BC of inlet or outlet; (3) the complex heart geometry has plenty of tiny features, which capturing needs great fine mesh creation in order to guarantee the robustness for the applied numerical scheme.

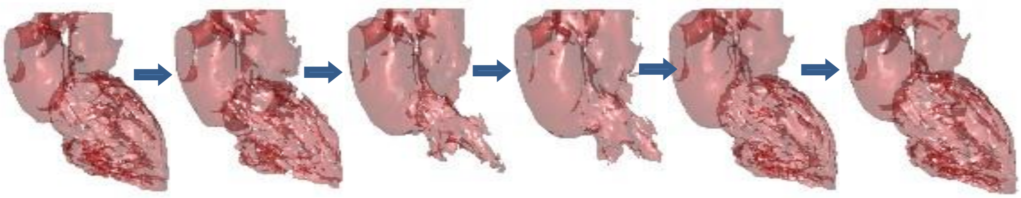


Fig. 1 The heartbeat schematic of shape changing in a heartbeat cycle

In recent years, the Computational Fluid Dynamics (CFD) simulation of the heart flow has made great progress and has been playing an important role in investigating the heart function. The most active CFD approaches to simulate the heart flow might be roughly classified as three types: geometry-prescribed CFD methods, fictitious Fluid-Structure Interaction (FSI) methods and realistic FSI methods (Cheng, 2005). The geometry-prescribed CFD method, simulating problems on prescribed moving meshes or boundaries constructed mostly from Computerized Tomography (CT) or Magnetic Resonance Imaging/Tomography (MRI/MRT) data (Baccani, 2003), (Keber , 2003), (Long, 2003), (Nakamura, 2002). However, almost all of them made some simplification when modeling the heart geometries and heartbeat process.

In this paper, a dynamically adaptive grid method based of the moving body in FlowVision is described, and combined with the automatic geometry replacement; enabling the dynamic simulation of the heartbeat process and modeling of the internal flow mechanism.

2. NUMERICAL MODELLING

The applied numerical schemes are integrated in the FlowVision software, which is based on the Cartesian grid and solving URANS equations. Important feature is that the dynamic mesh adaptation can be generated automatically, and it will reconstruct the mesh dynamically in real-time, especially interesting to be adapted to the dynamic geometry of the heartbeat, where solver automatically replaces the geometry at each time-variant change of the heart shape.

2.1 Finite Element Living Heart Model

Finite Element Living Heart Model (Figure 2) is an Abaqus project consisting two issued: electrical and mechanical modeling.

The electrical analysis is performed first and the resulting electric potentials are used as the excitation source for a subsequent mechanical analysis. The electrical and mechanical analysis use the same mesh topology. The electrical analysis begins at 70% ventricular diastole in the cardiac cycle. The electrical analysis is performed using the Heart Transfer procedure in Abaqus/Standart. Step time of electrical analysis is 500 ms.

The Mechanical analysis contains multiple steps of Dynamic, Explicit type: Pre-load, Beat1, Recovery1. On the Pre-load step is achieved the approximate pre-stressed state of the heart by linearly ramping up the pressure in the chambers. Step time is 0.3 s. On the Beat1 step - atrial and ventricular contraction phase of cardiac cycle during which voltages from the electrical analysis are applied. Step time is 0.5 s. And on the Recovery1 step – cardiac relaxation and ventricle filling phase. Step time is 0.5 s.

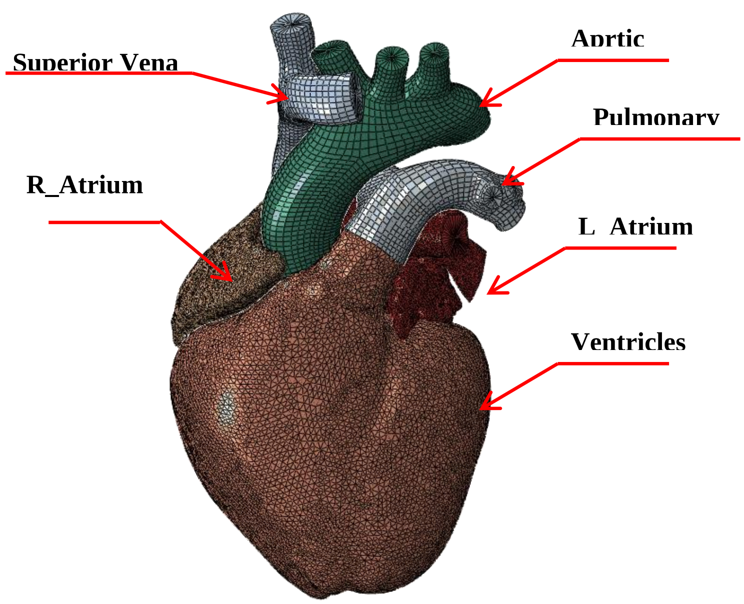


Fig. 2 Finite Element Living Heart Model

2.2 MRT Heart frames modeling

The heart model is a dynamic high-fidelity model of a normal (healthy), 4-chamber adult male human heart. It includes well-defined anatomic details of the heart as well as proximal vasculature, such as the aortic arch, pulmonary artery, and superior vena cava (SVC). Assuming the duration of a heartbeat cycle is 1s, which corresponds to a heart rate of 60 beats per minute, a typical adult (resting) heart rate.

Heart geometry frames originated from DICOM file, which obtained from CT scan. By inserting the set of DICOM files in Simpleware ScanIP software (480 slices in 1 frame), then the 100 heart frames (geometries) can be generated, as shown in Figure 3.

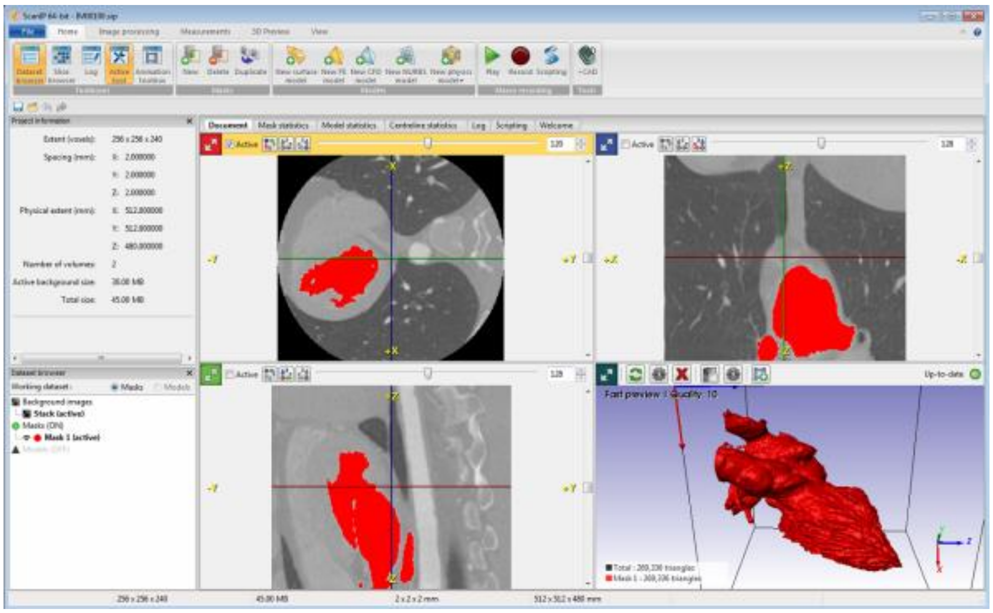


Figure 3 Heart frames generation from DICOM files in ScanIP

Using the scripting language available in Simpleware ScanIP software it was possible to automatize the process of generating 100 heart geometries. Since the shape of heart is very complex and includes many tiny features such as: gaps, overlaps, narrow regions, etc. which could possibly cause some problems in mesh generation it is essential to check and fix the geometries before CFD modeling. Quality check and cleaning was automatically executed in 3DTransVidia healing and repair software to make it applicable for FlowVision.

By using described workflow the heartbeat frames (geometries) representing heartbeat cycle can be automatically extracted, discretized and simulated in the time dependent way.

2.3 CFD Governing equations

Numerical simulation of blood flow in heart is based on solution full Navier-Stokes equations written for incompressible fluid. Continuity equation:

$$\vec{\nabla} \cdot \mathbf{V} = 0$$

Impulse conservation equation

$$\rho\left(\frac{\partial \mathbf{V}}{\partial t} + \left(\mathbf{V} \cdot \vec{\nabla}\right)\mathbf{V}\right) = -\vec{\nabla} P + \vec{\nabla} 2(\mu + \mu_t)\,\hat{\mathbf{e}}$$

$$\hat{\mathbf{e}} = \left\{e_{ij}\right\}; \quad e_{ij} = \frac{1}{2}\left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}\right)$$

Where t is time, $\rho, \mathbf{V}, P, \mu$ – density, velocity, pressure, viscosity of the blood. As blood flow in heart has high Reynolds number, k-ε turbulence model (Wilcox, 1994) is used to define μ_t - “turbulent” viscosity in Navier-Stokes equation:

$$\frac{\partial k}{\partial t} + \vec{\nabla}(\mathbf{V}k) = \frac{1}{\rho}\vec{\nabla}\left((\mu + \frac{\mu_t}{\sigma_k})\vec{\nabla} k\right) + \frac{G}{\rho} - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + \vec{\nabla}(\mathbf{V}\varepsilon) = \frac{1}{\rho}\vec{\nabla}\left((\mu + \frac{\mu_t}{\sigma_\varepsilon})\nabla \varepsilon\right) + \frac{\varepsilon}{k}(C_1 \frac{G}{\rho} - C_2 \varepsilon)$$

$$\mathrm{G} = \mu_t \mathrm{S}; \; \mu_t = C_\mu \rho \frac{k^2}{\varepsilon};$$

$$S = \frac{\partial V_i}{\partial x_j}\left\{\mu_t\left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}\right)\right\}$$

σ_k =1,0; σ_ε =1,3; C_μ =0,09; C_1 =1,44; C_2 =1,92

where k – turbulent energy, - turbulent enerfy dissipation

2.4 CFD model of Living Heart

Boundary condition for noth – MRT and LH model is shown in Figure 4 and consists in 3 boundary conditions – inlet, outlet and no-slip walls.

Living heart has too complex surface to use ordinary approaches in CFD for generation curvilinear meshes and using Lagrange approach for taking into account motion of the heart walls. FlowVision has full automatic mesh generator based on sub-grid geometry resolution method (SGGR) (Aksenov, 1998). SGGR is advanced analog of well known cut-cell method. In SGGR initially Cartesian mesh is introduced in computational domain, then cells crossed by boundaries of computational domain is cut but the surface and transformed in complex polyhedrons. To take into account motion of the heart walls, Euler approach is used. In this approach walls moves along stationary mesh, mesh cells could be appeared, disappeared and changed their shape and volumes. All governing equations approximated on this mesh is written for cell volumes that could be changed to take into account action of moving walls on the blood flow. The mesh is refined (adapted) using several conditions: to resolve fine structures of the heart surface and to resolve high gradients of the blood velocity to achieve sufficient accuracy of the numerical simulations. Mesh adaptation results in splitting mesh cell into 8 cells and smaller cells also could be split forming octree of the cells in computer memory. Blood flow is not steady state and during simulation in some regions, conditions for mesh refining could not be satisfied. In this case, process of making mesh more coarse is applied: previously refined cells are joined to form bigger cells to realize fully dynamic mesh adaptation.

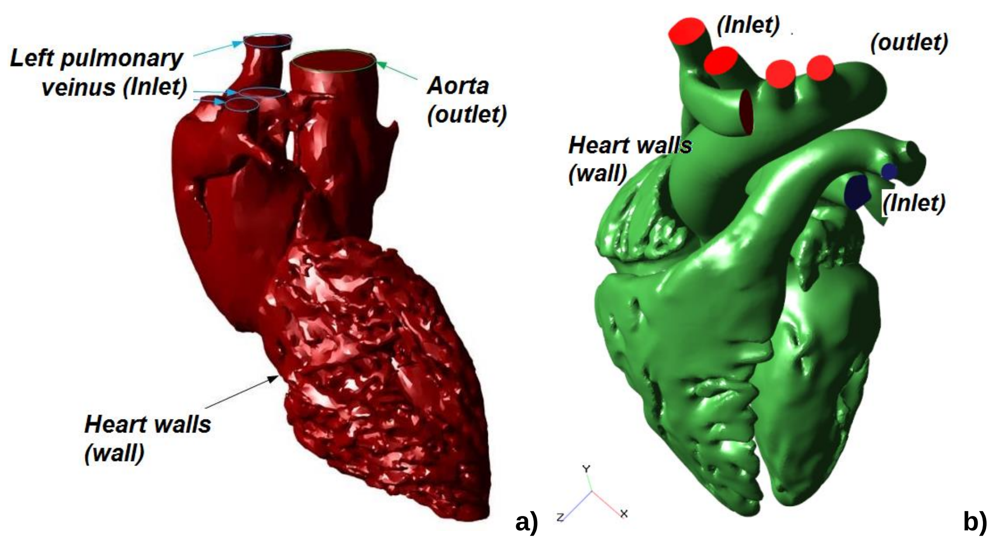


Figure 4. Boundaries of heart CFD model a) MRT model b) LH model

Dynamic mesh adaptation possibility is used for mesh convergence investigation. Simulating blood flow during heart beating is made for several mesh adaptation criteria to calculate dependence of some parameters on mesh refinement. And using this information a most coarse mesh is selected that has enough difference with fine mesh simulation. This selected mesh is presented in this paper.

With the heart shape changing in heartbeat process, it will also reconstruct the mesh dynamically in real-time, thus the total number of cell is dynamic, changing with the time and ranging from 2.0 to 3.2 million, as shown in Figure 5.

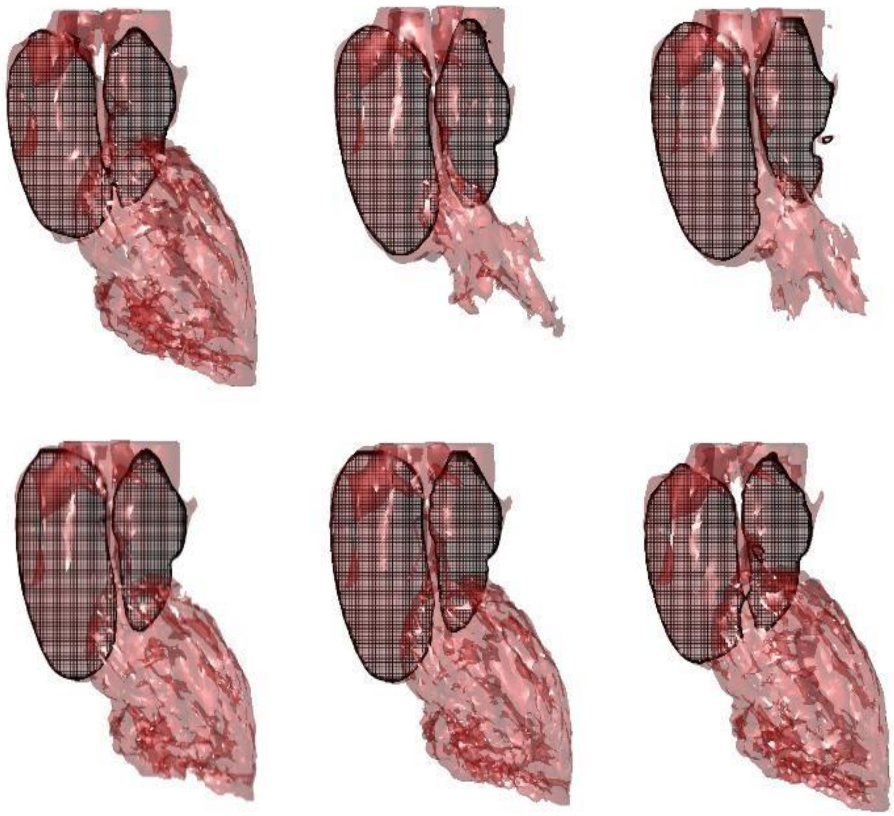


Figure 5. Mesh deformation within heartbeat process

2.5 Numerical setup

For the internal flow of heartbeat cycle, how to define the property of blood will have a big influence on the results. Blood viscosity is determined by plasma viscosity, hematocrit (volume fraction of red blood cell, which constitutes 99.9% of the cellular elements) and mechanical properties of red blood cells. Red blood cells have unique mechanical behavior, which can be discussed under the terms erythrocyte deformability and erythrocyte aggregation (Baskurt , 2003). Because of that, blood behaves as a non-Newtonian fluid. However, it is reasonable to regard it as a Newtonian fluid when modelling arteries with diameters larger than 1 mm (Noutchie, 2005).

Considering the size of heart ventricle, it is much bigger than 1mm, so we can use Newtonian fluid to modeling the blood flow inside the heart.

Assume the viscosity of blood is 0.0035 kg/(m·s) (i.e. 3.5 mPa·s). For big vessels as heart with high (turbulent) Reynolds number the blood can be assumed like Newtonian fluid. The numerical setup applied in FlowVision is based on solving URANS equations for all heartbeat process. Considering the flow medium is blood and low speed of the internal flow, it is reasonable to assume the flow is incompressible and without heat transfer. To simulate the turbulence, KES model (standard k-ε turbulence model) has been involved. In the macro-circulation, blood vessels are relatively large and the fluid velocity is relatively high, and therefore, the standard k-ε turbulence model is commonly used to solve for the flow field under these condition (Rubenstein, 2015).

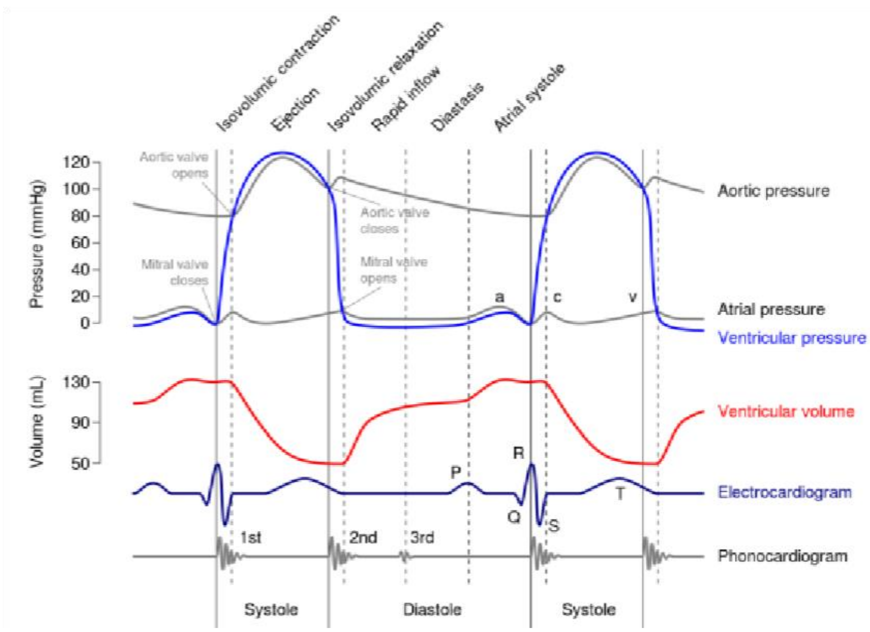


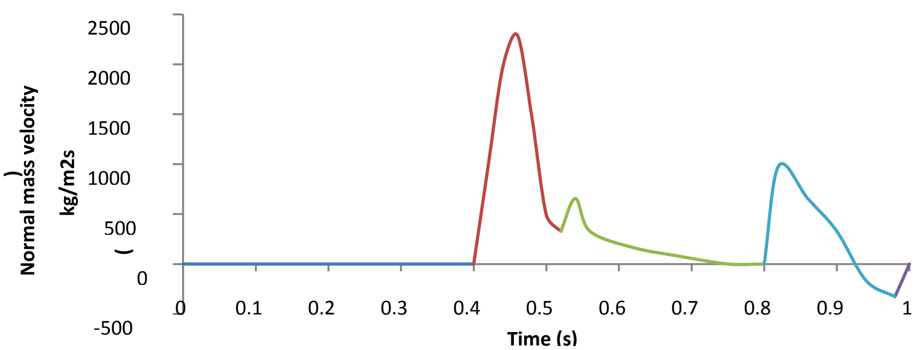
Figure 6 Wiggers Diagram

The parameters of the time-dependent boundary conditions are extracted from Wiggers Diagram (Figure 6), which is a standard diagram used in cardiac physiology. The cardiac cycle is the sequence of events that occurs when the heart beats. There are two phases of the cardiac cycle. In the diastole phase, the heart ventricles are relaxed and the heart fills with blood. In the systole phase, the ventricles contract and pump blood to the arteries. One cardiac cycle is completed when the heart fills with blood and the blood is pumped out of the heart.

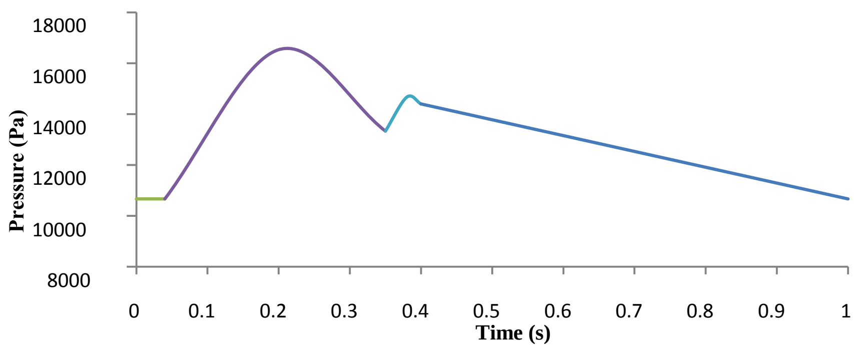
The aortic pressure can be used to define the pressure for outlet (Figure 7,b) and the normal mass velocity of the inlet can be deduced from the ventricular volume (Figure 7,a). When modelling this case, we assume the heart beat cycle is 1 sec. So for the normal mass velocity, it is systole period and there is no inflow for the inlet before 0.4s, so it is equal to 0 kg / (m^2·s). Afterwards, diastole period starts, the normal mass velocity isn't equal to zero, combined the volume rate and the area of inlet, the mass velocity of inlet can be obtained by using formula

$$\dot{m} = \frac{\rho}{S} \cdot \frac{\Delta V}{\Delta t}$$

where $\dot{m}$ is the mass velocity, ρ is the density, S is the area of inlet, V is the volume of ventricular and t is the time.



a)



b)

Figure 7. Boundary condition parameters:
a) Inlet normal mass velocity, b) Pressure at outlet

Time independency has been pursed for the unsteady simulation. Consequently, different time steps (0.01s, 0.005s, 0.002s, 0.001s and 0.0005s) have been tested to ensure time independent results. When the time step is bigger than 0.001s, the simulations are unstable and diverged around T=0.4s (from that the inlet mass velocity has a big change). For the time step 0.001s and 0.0005s, the simulations run stably and obtained similar results. Considering the results reliability and computational efficiency, the time step is selected as 0.001s.

2.6 Dynamic simulation with geometry automatic replacement

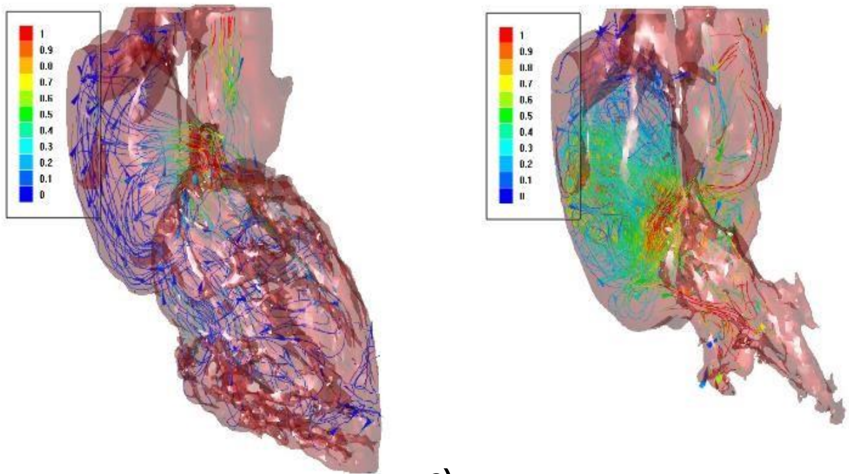
To simulate the heartbeat process, in which the heart shape is time-variant, 100 heart frames at different stage of heartbeat circulation are generated from CT scan images, and therefore they can keep even the small features as real human heart. By solving the URANS equations, and replacing the different heart frame at specific time step, the heartbeat process can be dynamically simulated in the time dependent way.

The heart frames are too many so that it is inappropriate to replace geometries manually. Thus some command lines in batch mode are predefined to achieve automatic geometry replacement. By using these command lines, 100 heart frames can be simulated one after another in chronological order, which ensures the heartbeat process goes smoothly and increases the robustness of calculation which might be effected by geometric data. Fig. 11 illustrates the mechanism of unsteady simulation with geometry automatic replacement. There are 10 time steps between the intervals of each 2 adjacent frames. Before each frame replacement, it will automatically save the results.

3. RESULTS ANALYSIS

Dynamic simulation was achieved for the heartbeat process (in one complete heartbeat). The dynamic simulation recovers the main characteristics of the real heart internal flow, Fig. 8-10 illustrate the internal flow status at different time. From a general view, the flow is unsteady and accompany with much vortex flow, which increase the complexity for flow mechanism analysis.

A blood flow speed reached the maximum proximately at t=0.2s, which corresponds the peak pressure in Figure 7,b. In a cardiac cycle, the blood flow speed is extremely slow (Figure 8), the magnitude is always ranging from 0 to 1m/s. The average velocity during systole is also bigger than diastole period, and the peak velocity at different stage is agreed with the results in reference literature (Wexler , 1968).



a) b)

Figure 8. The streamlines of blood in MRT model a) t= 0.8s b) t=1.0s

Generally in the human body, blood flow is laminar. (a) t= 0s (b) t=0.2s However, under conditions of high flow, particularly in the ascending aorta, laminar flow can be disrupted and become turbulent, as shown in Figure 8-10 both for MRT and LH models. In a cardiac cycle, the turbulence viscosity in the aorta is always far bigger than the viscosity in laminar flow. Even in most other part of the heart, the blood flow behaves as turbulent flow. Thus the internal flow in human heart can be regarded as fully turbulent state.

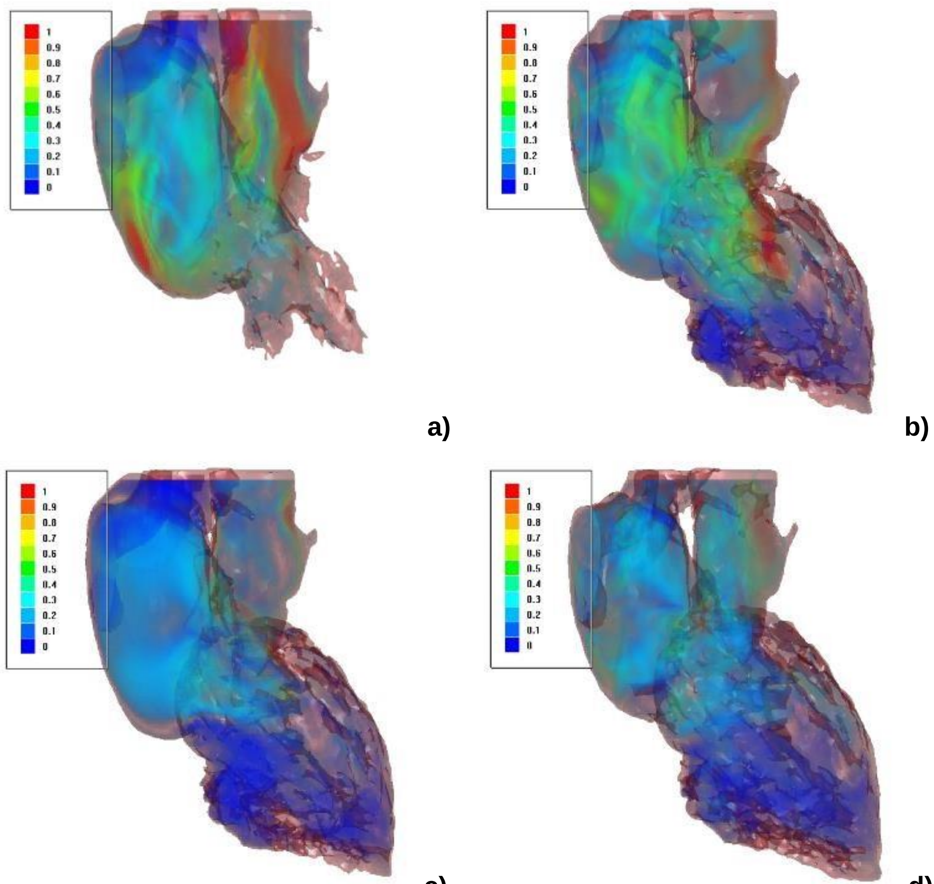


Figure 9. Volume visualization of velocity distribution in MRT model, time is a) 0.4s, b) 0.6 s, c) 0.8 d) 1.0s

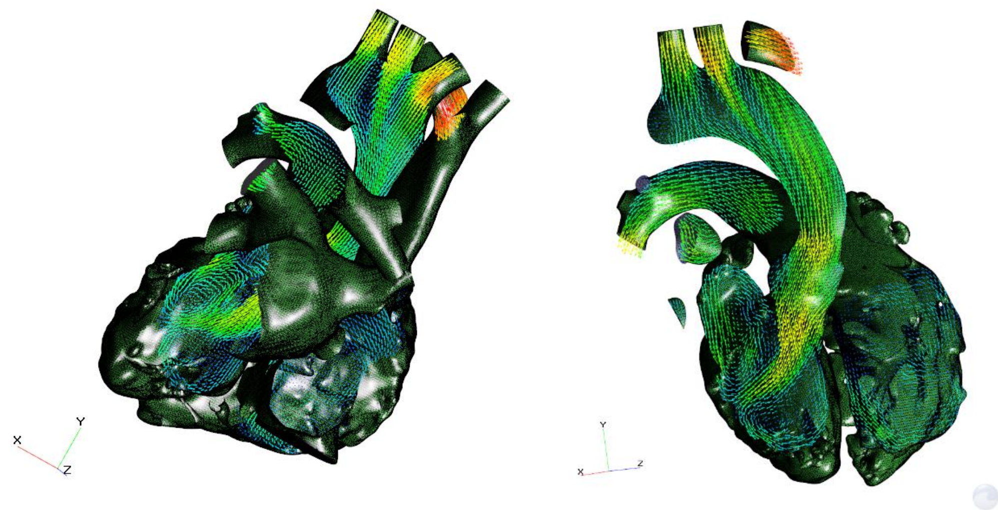


Figure 10. Blood velocity distribution in FE LVH model

4. CONCLUSION

In this paper, a novel approach for simulating blood flow using data of MRI/MRT scanner and Finite Element Living Heart Model and CFD tool FlowVision is presented. Sub-Grid Geometry Resolution method for mesh generation with Euler approach for taking into account moving walls of living heart used in FlowVision allows to get pressure and velocity distribution of blood inside the heart as well as with visualization of these fields on base of planar and volume rendering.

As the early research work of heart modeling, the internal flow simulated in this paper didn't involve the influence of cardiac valves, and this will be carried out in future work, which will focus on FSI analysis, combining with electromagnetic prompting valves.

5. REFERENCES

1. A Kumar, "Changing trends of cardiovascular risk factors among Indians: a review of emerging risks," Asian Pacific Journal of Tropical Biomedicine, 2014, 4(12): 1001-1008.
2. Baskurt OK, Meiselman HJ, "Blood rheology and hemodynamics". Seminars in Thrombosis and Haemostasis 29: 435–450, 2003
3. 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.
4. David Rubenstein, Wei Yin, Mary D. Frame, "Biofluid Mechanics: An Introduction to Fluid Mechanics", London: Elsevier, second edition 2015, pp. 466.

5. Cheng Y, Oertel H, Schenkel T, “Fluid-structure coupled CFD simulation of the left ventricular flow during filling phase,” Annals of biomedical engineering, 2005, 33(5): 567-576.
6. Baccani, B., F. Domenichini, G. Pedrizzetti, “Model and influence of mitral valve opening during the left ventricular filling,” J. Biomech. 36:355–361, 2003.
7. Keber, R, “Computational fluid dynamics simulation of human left ventricular flow,” PhD Dissertation, University of Karlsruhe, Karlsruhe, 2003.
8. Long, Q., R. Merrifield, G. Z. Yang, X. Y. Xu, P. J. Kilner, D. N. Firman, “The influence of inflow boundary conditions on intra left ventricle flow predictions,” J. Biomech. Eng. 125:922–927, 2003.
9. Nakamura, M., S. Wada, T. Mikami, A. Kitabatake, T. Karino, “A computational fluid mechanical study on the effects of opening and closing of the mitral orifice on a transmitral flow velocity profile and an early diastolic intraventricular flow.” JSME Int. J. Ser. C – Mech. Syst. Mach. Elem. Manufact. 45:913–922, 2002.
10. Wexler L, Bergel DH, Gabe IT, “Velocity of blood flow in normal human venae cavae,” Circulation Research, 1968, 23 (3): 349-359.
11. Wilcox D. C. (1994) "Turbulence modeling for CFD", DCW Industries, Inc., 460 p.
12. 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.

.