PATIENT-SPECIFIC COMPUTATIONAL FLUID DYNAMIC SIMULATION OF INTRAVENTRICULAR HEMODYNAMICS: INTRODUCING MITRAL VALVE MOTION AS PRESCRIBED BOUNDARY CONDITION

Alessandra M. Bavo, IbiTech-bioMMeda, iMinds Medical IT, Ghent University, Belgium.
alessandra.bavo@ugent.be
Alison M. Pouch, Gorman Cardiovascular Research Group, University of Pennsylvania, PA
Joris DeGroote, Dep. Flow, Heat and Combustion Mechanics, Ghent University, Belgium
Jan Vierendeels, Dep. Flow, Heat and Combustion Mechanics, Ghent University, Belgium
Joseph H. Gorman, Gorman Cardiovascular Research Group, University of Pennsylvania, PA
Robert C. Gorman, Gorman Cardiovascular Research Group, University of Pennsylvania, PA
Patrick Segers, IbiTech-bioMMeda, iMinds Medical IT, Ghent University, Belgium

Key Words: Mitral valve, CFD simulation with moving boundaries, ultrasound images

Previous research on intraventricular flow fields using computational fluid dynamics (CFD) have mainly made use of the prescribed geometry approach, imposing the motion of the left ventricle as a boundary condition. In a few cases, the orifice of the mitral valve (MV) was included to account for the influence of the valve, but the leaflets themselves were, to the best of our knowledge, ignored in such approaches. The novelty of this study is the introduction of the movement of the entire, patient-specific MV in intraventricular CFD simulations, with this initial report focusing on the valvular motion. The main challenge resides in the transfer of the large displacement of the valve from the images to the CFD simulations, preserving a good quality of the valvular surface and of the fluid domain grid. The *a priori* knowledge of the valve position is available from real-time 3D ultrasound (US) images acquired during the cardiac cycle with a temporal resolution of 13 images/second. The MV was segmented using a previously described method that combines multi-atlas label fusion and deformable medial modeling to obtain volumetric geometrical models of the anterior and posterior leaflets. The computational fluid grid was realized in PyFormex, a python-based code which allows for the manipulation of complex geometries. The 3D, time-dependent position of the valve is transferred to the fluid solver with a natural spline interpolation scheme to obtain a smooth wall boundary movement. The spatial correspondence between the US reference and the computational moving surfaces is managed within the flow solver, by introducing an interpolation function based on the three nearest neighbors. The calculation is performed in Fluent (Ansys), solving the Navier-Stokes equations on a deforming mesh using the arbitrary Lagrange-Eulerian formulation.

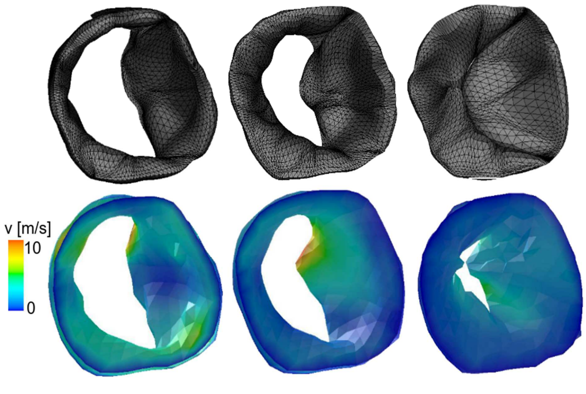

Figure 1: ultrasound images (upper panel), computed valve configuration (lower panel), wall velocity.

Figure 1 displays the motion of the valve at three time-points during the closing phase of the valve, comparing the US image to the corresponding CFD configuration. The closed configuration of the MV is the most challenging: a contact constraint has to be introduced to avoid the overlap of the leaflets, and a small gap between the coapting leaflets is permitted to avoid the division of the fluid domain. An arbitrary artificial porosity can be added in this area to reduce the backflow. From a computational point of view, the prescribed geometry approach is very convenient: with the valve position known *a priori,* a time-consuming and complex FSI simulation is not strictly necessary, and the reliability of the motion is guaranteed by the use of the patient-specific source data. The inclusion of the left ventricle to analyze the intraventicular flow is currently under investigation. The development of such a model can provide a tool to simulate clinical cases and predict the hemodynamic field in the region, which will possibly provide the clinicians extra insights about the intraventricular flow and valvular dynamics, in particular to estimate the outcomes of surgical interventions.