COMPUTATIONAL FLUID DYNAMICS APPROACH TO THE CONTACT PROBLEM IN SIMULATIONS OF FLEXIBLE LEAFLET HEART VALVES

Alessandra M. Bavo[1], Liesbeth Taelman[1], Joris Degroote[2] , Jan Vierendeels[2], Patrick Segers[1]
[1] IBiTech-bioMMeda – Ghent University, Ghent, Belgium; [2] Department of Flow, Heat and Combustion Mechanics – Ghent University, Ghent, Belgium.

Introduction

When simulating the kinematics and fluid mechanics of a cardiac valve, an important aspect is the modelling of the contact between different solid surfaces, preventing a backflow during the closing of the valve (regurgitation). The numerical description of this contact problem is, however, challenging. In this work, the contact is studied in a computational fluid dynamic (CFD) simulation, as a preliminary step preceding the fluid-structure interaction (FSI) simulations.

Methods

The software Fluent is used for all the simulations. A two-dimensional, axisymmetric geometry is used, as shown in Figure 1.

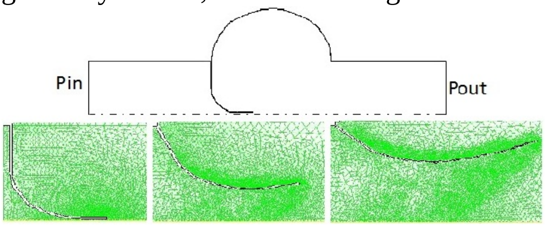

Figure 1: upper panel: 2D axisymmetric geometry, lower panel: the leaflet in 3 different positions (closed, opening, fully opened)

As boundary conditions, a sinusoidal pressure with an amplitude of 75 mmHg is imposed at the inlet, while the pressure is kept constant at the outlet. In the preliminary CFD simulations, the movement of the leaflet is prescribed by an analytical function. The Navier-Stokes (NS) equations are solved with an arbitrary Lagrangian-Eulerian (ALE) formulation, and a spring model is used to manage the movement of the fluid domain. As such, a high quality grid is obtained throughout the simulation, moving coherently with the leaflet.

When the leaflet approaches the axis of symmetry, the movement of the leaflet is hampered and a thin layer of cells belonging to the fluid domain is preserved, avoiding the splitting of the domain. To reduce the backflow (i.e. simulate contact) a change in the porosity characteristics of the medium is imposed in this layer of cells. The presence of a porous zone is modelled by adding a momentum source term in the NS equation.

Results

The simulations are performed with 4 different values of porosity resistance during the contact phase. The average of retrograde flow is negligible for high values of porous resistance, as listed in Table 1.

Table 1: Percentage of backward flow, with no porous zone (NP) and normalized porosity coefficients equal to 1, 10, 100.

	Resistance [1/m^2]	**% backflow**
NP	---	0.092
PR1	1	0.046
PR2	10	0.0186
PR3	100	0.0066

Figure 2 depicts the flow during one cycle. It can be seen that the backflow (PR2, PR3) in the closed phase is limited, even if a relevant pressure drop across the valve is present.

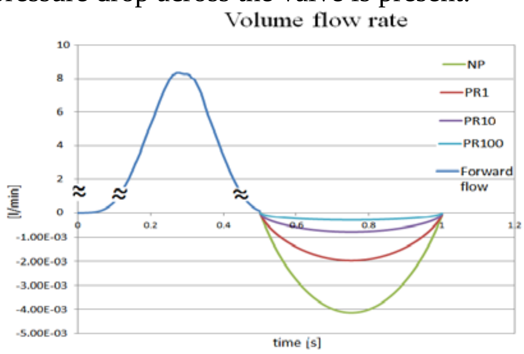

Figure 2: Volume flow rate in a cycle

Discussion

By changing the porous characteristics of the fluid zone, it is possible to simulate a contact phenomenon in a CFD simulation without splitting the fluid domain. During the open phase the flow is not affected, while the backflow becomes negligible during the closed phase. This technique will be adopted in the fully coupled FSI problem of more realistic valve geometries.