Experimental and Numerical Design and Investigation of Flow Field in a Three-Dimensional Artificial Heart Model

N. Filipovic\textsuperscript{1}, M. Kojic\textsuperscript{2} and M. Radosavljevic\textsuperscript{3}
\textsuperscript{1}Technical Faculty, Cacak/ Serbia-Yugoslavia
\textsuperscript{2}Faculty of Mechanical Engineering, Kragujevac/Serbia-Yugoslavia
\textsuperscript{3}High Technical School, Kragujevac/Serbia-Yugoslavia

Introduction

Artificial heart devices are required to satisfy high functionality and bio-compatibility demands for successful use. Although the experimental studies play an important role in the design of artificial heart devices, they can provide flow characteristics for limited regions of the flow field. Detailed knowledge of the flow quantities can help the design engineer to improve the artificial heart. Efforts in the development of a modern generation of these devices are supported by the application of the advanced Computational Fluid Dynamics (CFD) methods presented in this study.

Methods

We presented experimental model of the artificial heart device which is completely controlled by a personal computer. The input and output variables (velocity and pressure) are obtained by measurements. This paper also includes an extension of CFD techniques to the artificial heart flow simulation. We have implemented a numerical procedure based on the Arbitrary Lagrangian-Eulerian (ALE) approach for the fluid-structure interaction., (Filipovic, 1999), (Kojic and Bathe, to be published). The unsteady three-dimensional Navier-Stokes equations are solved for the fluid domain.

The geometry of the artificial heart model is composed of a cylindrical chamber with two tubed (1cm diameter). The lower base of the cylindrical chamber is an elastic membrane which is moving against the piston mechanism by equation $m\ddot{x} + cx = F(t)$. This motion is governed by the computational study, with the Reynolds number based on the tube diameter and velocity equals 850cm/s. The pumping action is realized by the piston which pushes the membrane up and down. The diameter of the bases of cylindrical chamber is 6 cm and the height is 3cm.

In order to handle the geometrical complexity and the moving boundary problems, the ALE method is employed. The ALE method allows a great flexibility when the boundary motion creates large displacements. The computational grid for this heart model is shown in Fig. 1a.
The inflow (mitral) and outflow (aortic) tubes contain the virtual tilting disks that open and close to act as the valves. The mitral valve open and close with a phase difference. These data were obtained by fitting the experimental results.

Figure 2: Experimental setup scheme for examining flow through the artificial carotid artery bifurcation by using the artificial heart device (pump)
Results & Discussion

Fig. 2 shows the experimental setup for flow through the artificial carotid artery bifurcation. The maximum flow in experiment was 2.0 lit/min and the maximum pressure in the system was 160 mmHg. The photography of the artificial heart device is shown in Fig. 1b.

The numerical results are presented to demonstrate how the CFD can be used to model the flow in the artificial heart. A recirculation region can be seen at the connection between the chamber and the inflow tube. The velocity vectors at $t/T=0.4$ are shown in Fig. 3a when the elastic membrane moves in the downward direction and the mitral valve is open. Figure 3b shows the numerical velocity vectors at $t/T=0.7$. At this time the elastic membrane is moving the upward direction and the aortic valve is open.

The comparison of the computational results with the experimental data show a good agreement. Flow investigations were carried out experimentally and numerically on the automated system controlled by a personal computer. This application and preliminary results offer a basis to explore a much broader design options than the traditional parametric methods. This complex numerical tool can assist developers of the artificial heart device in to design the blood-wetted components that minimize thrombosis and hemolysis, while simultaneously provide a maximum flow performances.

![Figure 3a: Vector distribution inside of the chamber for $t/T=0.4$.](image1)

**Figure 3b:** Vector distribution inside of the chamber for $t/T=0.7$.

References