Numerical simulation of the turbulent flow field distal to an aortic heart valve

YOS MORSI¹,*; AMAL AHMAD¹ and AMR HASSAN²

¹ Bio-fluid Dynamics Group, Centre for Modelling and Process Simulation School of Engineering and Science, Swinburne University of Technology, Hawthorn, Victoria 3122, Australia
² School of Mechanical, Materials, Manufacturing, Engineering and Management, Division of Mechanical Engineering, Nottingham University, Nottingham NG7 2RD, UK

Received 17 November 1999; revised 4 April 2000; accepted 18 April 2000

Abstract—A numerical technique is described which simulates the turbulent flow through a aortic Jellyfish valve, for two volumetric flow rates of 15 and 26 l/min representing peak systole flow. The steady incompressible Navier–Stokes equations written in three-dimensional (3-D) format are solved iteratively using a computational fluid dynamics code (Bio-FL). The numerical results show that the flow at the edge of the membrane of the valve splits into two nearly symmetrical jets with similar phenomenological features. Moreover, the 3-D flow simulations indicate the existence of two vortices in the immediate vicinity of the valve ring. Although these vortices are attenuated rapidly downstream by diffusion, they can have an adverse effect on erythrocytes and active platelets. It is also shown that elevated shear stresses occur in the vicinity of the upstream surface of the open occluder. In general, the numerical predictions compare well with the experimental measurements made at various locations downstream of the valve. The locations and the values of maximum velocity and shear stress, as well as width and length of re-circulation regions, are correctly predicted.

Key words: Prosthetic aortic valves; numerical simulation; spiral vortices; systolic steady blood flow.

1. INTRODUCTION

Research and development effort toward the evolution of bio-compatible, hydrodynamically efficient and structurally sound prosthetic heart valves has been a continuing process of development since the first implantation of a prosthetic valve in 1954 as viable means of prolonging and improving the lives of patients in need of cardiac support. There are many problems associated with artificial heart valves and the solution of those linked to the hydrodynamic performance of valves is considered to

*To whom correspondence should be addressed. E-mail: ymorsi@swin.edu.au
be essential in assessing the suitability of different valves for clinical use. The two major problems related to blood flow are thromboembolism and hemolysis [1–3].

To estimate the detrimental fluid forces that limit the use of artificial devices, it is necessary to determine the flow field through them. In principle, the analysis of fluid dynamics of artificial devices can be carried out either experimentally or numerically, or by using a combination of both techniques. However, despite the fact that many formidable fluid dynamics problems have been tackled successfully using computational fluid dynamics (CFD) almost all haemodynamic evaluations of prosthetic heart valves to date have been obtained through in vitro and in vivo experimentation, e.g. [4–7].

Although they can offer considerable detailed information on the flow characteristics of artificial heart devices, experimental investigations are limited in their ability to provide a three-dimensional (3-D) picture showing the complete nature of the flow. They can be very expensive and time consuming, and offer little flexibility in changing operating and geometrical conditions. Moreover, accurate experimental measurements are difficult to obtain because of the moving boundaries in the artificial devices. In contrast, numerical simulations can provide an accurate whole field solution to the problem and can be used to evaluate the suitability of artificial devices under various operating conditions at the very early stage of their development.

It is well known that elevated shear stress and flow separation regions around the valve are the cause of red blood cell damage and platelets [1, 3, 8]. From this standpoint, detailed 3-D knowledge of the flow quantities can help the designer improve the geometry of artificial heart devices where smooth blood flow characteristics are required. This information can be readily obtained using numerical simulation techniques.

Encouraged by the recent advances in CFD technology, various researchers have investigated the hemodynamic characteristics of many common artificial valves. The numerical simulation of the flow characteristics of prosthetic heart valves in the natural heart is worthy of mention. This work, [8], included the analysis of the opening and closing performance of the valve and the elastic behavior of the walls. However, only laminar two-dimensional (2-D) flow conditions were investigated. The same research group used the combined vortex-grid method to analyze the 2-D blood flow behavior through a mitral valve and demonstrated the capability of modeling the elastic behavior of heart muscles [9, 10]. Similar 2-D numerical simulations of turbulent steady flow through five trileaflet aortic heart valves have also been reported [11].

In addition to the above 2-D flow investigations, there are a number of 3-D numerical studies related to the analysis of the flow behavior around artificial devices reported in the literature. Notable early research in this area was the successful simulation of the flow characteristics through the left ventricle and a mitral valve, [12, 13]. Later the 3-D steady flow fields around an aortic Bjork–Shiley valve were reported [14] and it was noted that the valve generated