Numerical Simulation of Transient Blood Flow in the Socket Region of a Bi-leaflet Heart Valve

Abstract

A computational fluid dynamics analysis of the On-X® bi-leaflet heart valve has been conducted. Unsteady blood flow in the hinge area was investigated using STORM/CFD2000. The Navier-Stokes equations were solved using a finite volume grid, and a PISO (pressure-implicit with splitting of operators) algorithm was employed to compute the pressure-velocity coupling. An alternating direction implicit (ADI) scheme was used to solve the set of linear algebraic equations. A moving grid methodology with a prescribed periodic motion was employed to simulate the opening and closing of the valve leaflet.

Particular attention was paid to simulating the local flow inside the hinge socket of the On-X® valve where experimental measurements are very difficult or impossible. The results show that local flow patterns and pressure distributions in the hinge area of the valve induce full wash-out during the opening and closing phases.

Introduction

The use of prosthetic mechanical heart valves to replace diseased or defective natural heart valves has been a common clinical practice for over thirty years. Since the first successful prosthetic heart valve implantation in the early 1960s, various types of prosthetic mechanical heart valves have been introduced and widely used for replacement surgery

Major clinical complications associated with prosthetic heart valves include hemolysis, thrombus formation, thromboembolism and fibrous tissue overgrowth. These complications are closely related to the *in vivo* hemodynamic behavior of a valve design. Individual valve designs produce characteristic local flow patterns in the vicinity of valve occluders. These patterns include disturbed or turbulent flow, flow separation, re-circulation, vortices, regurgitation jets and cavitation — all of which may damage blood elements and trigger the formation of thrombus. In addition, re-circulation, flow separation and flow stasis can also encourage the deposition of platelets and blood cells forming platelet aggregates.

Quantitative *in vitro* experimental studies of both upstream and downstream flow structures from various mechanical prostheses have been conducted over the course of the past two decades. Due to the limitations of experimental apparatus and the complex structural designs of mechanical prostheses, it is impractical to obtain detailed local flow structural information in the vicinity of the heart valve body, inside the valve orifice and inside the valve hinge areas. This information is very important for evaluation of hemodynamic behavior in the valve prior to clinical implantation.

Numerical simulations with appropriate experimentally determined physical boundary conditions can provide the necessary detailed level of understanding of the local flow characteristics of given valve design at virtually every point inside the valve orifice. The current work uses the techniques of computational fluid dynamics (CFD) to simulate the local flow pattern in a dynamically operating heart valve.

Computational Fluid Dynamics Model

Valve Geometry

The On-X® bi-leaflet heart valve consists of a valve orifice and two leaflets. The leaflets are mounted in the orifice through a specially designed hinge geometry, which opens and closes periodically in response to pressure differences across the valve. Details of the modeled hinge geometry are shown in Figure 1. Due to the inherent symmetry of the valve, computational modeling of just half of the valve volume was performed. The leaflet is modeled as a moving surface.

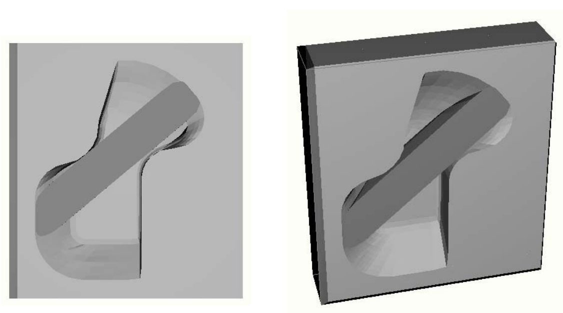

Figure 1. CFD model, hinge geometry.

Computational Grid

A particular effort was made to design a computational grid that would not be subject to tangling of the cell edges or to undue skewing or stretching of individual cell volumes as the grid moves in response to the leaflet motion. To better control the grid motion, the simulated domain was divided into several sub-volumes. Within each sub-volume, a power law distribution of cell center locations was used to optimize the grid arrangement. Altogether, a total of 21,420 cells were used for the simulation. STORM/CFD2000 allows grid points (nodes) to "slide" along a pre-defined surface or curve. This featured guarantees that the valve geometry remained unchanged while the leaflet was moving.

Boundary Conditions and Numerical Method

No slip wall boundary conditions were used for all external surfaces, while the internal solid regions, such as the valve orifice and the leaflet, were treated as internal obstacles with impermeable boundary surfaces. A time-varying velocity profile was prescribed at upstream inlet, driven by a mass flow rate that was based on experimental measurements. Pressure was prescribed at a fixed value at the downstream outlet, since only the relative pressure between the inlet and the outlet is relevant to the problem under consideration. A free-slip symmetry boundary condition is set on the symmetry plane in the axial direction. The unsteady blood flow in the hinge area is considered as a three-dimensional transient problem. The Navier-Stokes equations were solved using a finite volume method with a pressure-based algorithm for continuity and PISO algorithm for pressure-velocity coupling. A second-order accurate hybrid scheme was used for time marching. The flow is initiated with zero velocity. A fixed time step of 1.0×10^{5} sec was used for all numerical calculations.

Computational Results

Velocity and Pressure Fields

For the purpose of this discussion, the hinge area is divided into four sub-areas, identified as A, B, C and D in Figure 2.1. Three planar cuts and one surface cut are used to reveal the velocity and pressure fields of the hinge area flow (Figure 2.2). The cuts are made outside the socket, directly on the hinge flat and halfway into the socket, respectively. The surface cut is made in the socket bottom-gap. Arrows are used to represent the local velocity vectors, and a color scheme is used to indicate the fluid pressures.

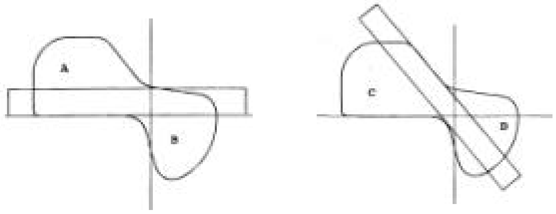

Figure 2.1. Sub-regions for hinge area.

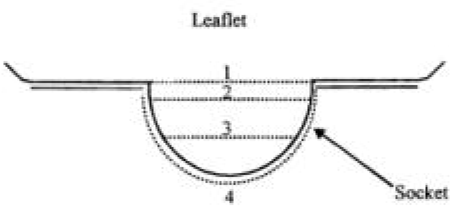

Figure 2.2. Cut locations in socket area.

Figures 3.1-3.4 shows the velocity and pressure fields on the out-of-socket plane at four characteristic moments during the cycle, starting with the leaflet in a fully closed position. Initially, the pressure field is fairly flat, but quickly builds as the flow is impinges against the closed leaflet. Once the leaflet begins to open, the flow is dominated by the large-scale orifice flow in the valve. As the pressure continues to increase in the left atrial chamber (the upstream side), the leaflet moves downstream with the flow and begins to rotate open. The flow varies smoothly without generating any vortices during the leaflet-opening process. When the pressure on the ventricular chamber side is higher than that in the atrial chamber, the velocity field reverses and the leaflet is driven to close. Two vortices develop on either side of the leaflet in response to flow inertia when the leaflet closes completely.

When the orifice flow enters the socket, it becomes locally restricted by the socket geometry. On the other hand, the socket flow merges into the orifice flow when it is out of the socket. The hinge flat cut just lies in the transient area of the two flows, showing characteristics of these flows, as shown in Figs. 4.1-4.4.

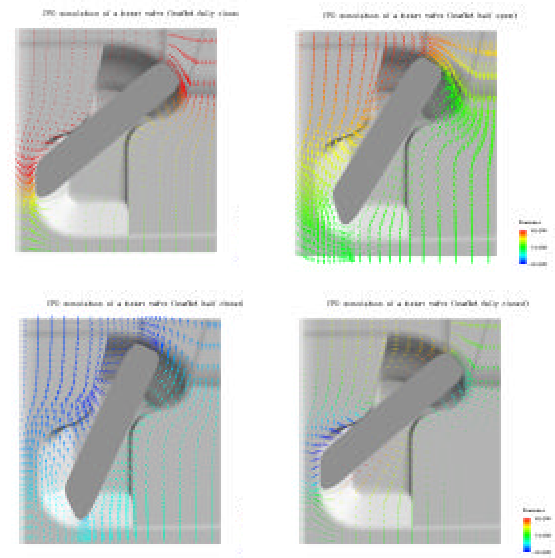

Figure 3.1-3.4. Pressure and velocity - location 1.

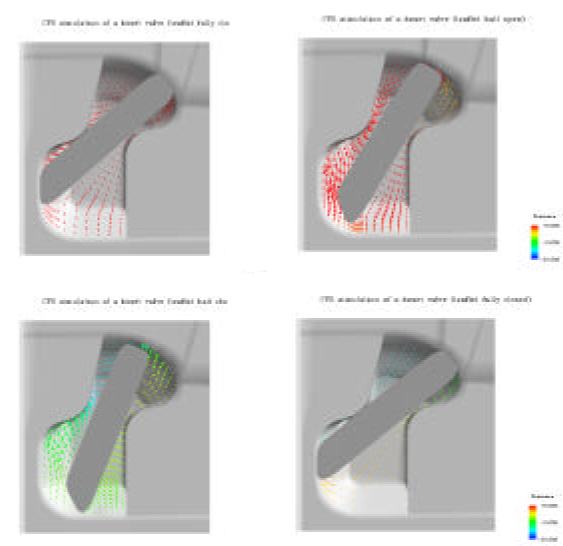

Figure 4.1-4.4. Pressure and velocity - location 2.

The influence of the socket sidewall on the flow can be clearly seen in area B, which is surrounded by a socket wall and a leaflet surface. When the leaflet starts to open, the pressure in the area drops and fluid is sucked into the socket and a vortex develops. However, once the leaflet is wide open, the sucking effect vanishes and the vortex disappears. When the orifice flow reverses, the leaflet begins to close, thereby squeezing the local socket fluid back into the orifice flow. This exchange repeats in the next cycle.

Two cuts are made inside the socket to understand the details of the local blood flow. One cut is made at a point halfway between the hinge flat and the bottom of the socket as shown in Figures 5.1-5.4. The flow in this plane is similar to that in the out-of-socket planes, except that the local flow in area D is somewhat stronger. The wall alters the local flow and forms a jet-like flow toward area D during the leaflet-closing process. The local flow in area B retains its low pressure during the opening process and sucks fluid in. It also maintains high pressure to squeeze fluid out during the closing process.

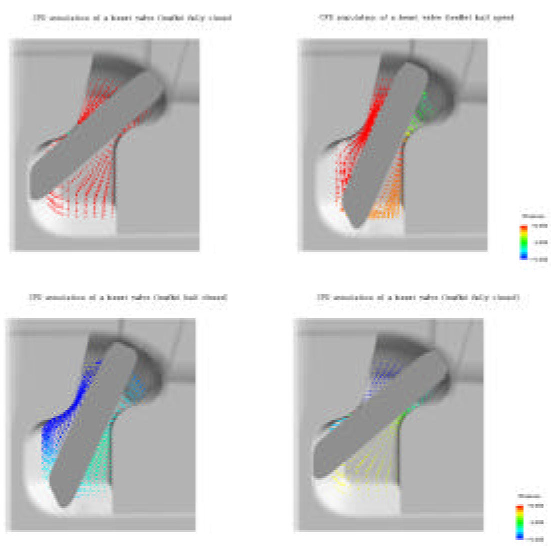

Figure 5.1-5.4. Pressure and velocity - location 3.

Figures 6.1-6.4 show the instantaneous streamlines within the heart valve at five different instances during the valve operating cycle. These plots indicate that the flow motions in the hinge area are in fact three-dimensional, with some streamlines passing over and other passing under the leaflet. These observations are consistent with the flow characteristics described in the previous section.

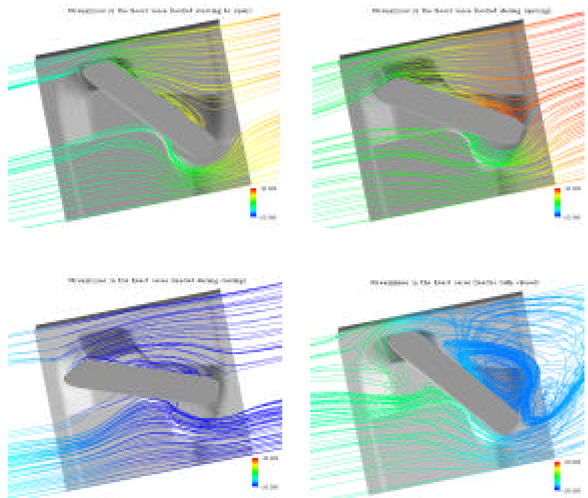

Figure 6.1-6.4. Streamlines.

Discussion and Conclusion

The three-dimensional unsteady blood flow in the hinge area of the On-X® bi-leaflet heart valve has been numerically simulated by solving the full Navier-Stokes equations with a finite volume method and moving grid technique. The simulation reveals the flow in the hinge area, where experimental measurements are very difficult or impossible. The overall results show that local flow patterns and pressure distributions in the hinge area of the valve induce full wash-out during the opening and closing phases, which suggest that the On-X® valve will not support mechanically induced thrombotic activities.