EXPERIMENTAL AND NUMERICAL ANALYSES OF THE STEADY FLOW FIELD AROUND AN AORTIC BJÖRK–SHILEY STANDARD VALVE PROSTHESIS

M. LEI,* A. A. VAN STEENHOVEN and D. H. VAN CAMPEN

Department of Mechanical Engineering, Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, The Netherlands

Abstract—To develop a numerical method for the description of the flow field around a Björk–Shiley (BS) standard valve prosthesis in aortic position, detailed experimental measurements and numerical calculations are performed under steady flow conditions. The experiment was conducted at Reynolds numbers up to 800. In order to perform LDA measurement of velocity in the vicinity of the valve with a curved sinus boundary, a mixture of oil and kerosine was used as the fluid which exactly matches the refractive index of the perspex aortic model. The velocity profiles at six positions in the vicinity and downstream of the valve were measured, including both axial and radial velocity components. The results show very clearly the existence of two nearly symmetric spiral vortex streams downstream of the valve. There is no recirculation area in the aorta downstream and also no obvious stagnation area in the minor orifice region near the valve when Re ≤ 800. Theoretically, the flow field of a BS valve is simulated by the flow pattern around a circular plate with an angle of incidence to the approaching stream. The numerical calculations were carried out by means of a 2-D model using the FEM together with the penalty function method. The maximum Reynolds number is 700. The results agree with the experimental results in the plane of symmetry when the Reynolds number is small. However, as the Reynolds number increases, the difference becomes evident. Our conclusion is that the steady flow field of a BS valve is completely 3-dimensional, featured by two spiral vortices. It cannot be simulated exactly by 2-D numerical calculations. To get more detailed and complete information about the flow field of this valve, 3-D numerical calculations are needed.

INTRODUCTION

Natural heart valves function mainly as one-way valves in vivo which are purely fluid-dynamical devices in nature (Lei and Kang, 1986). Therefore, the hemodynamical performance is very important in the quality evaluation of artificial heart valves. The artificial heart valve has continuously been in improvement ever since it was first successfully introduced into cardiac surgery in 1953 by Hufnagel and Harvey (see Wieting, 1969), but its hemodynamical performance has never reached the level comparable to the natural valves. Still many problems exist. For mechanical valve prostheses, thrombus formation and hemolysis are still very severe, due to which the patients implanted with mechanical valves need anticoagulant therapy all the time. Since thrombus formation and hemolysis are related to the flow pattern (Yoganathan et al., 1978), the study of the flow field around a prosthetic valve is very necessary in order to improve the valve design.

The Björk–Shiley valve prosthesis is one of the most popularly used mechanical valves in the world. Due to its special structure, it produces a more complicated flow field than other types of valve prostheses, which is very difficult to analyse theoretically. To get information about the flow field, both experimental and numerical methods have been used. For experimental studies, almost all the measuring techniques available have been used, such as laser Doppler anemometry (LDA) (Yoganathan et al., 1978, 1979; Khalighi et al., 1983; Chandran et al., 1985; Hanle et al., 1989; Tiederman et al., 1986; Figliola and Mueller, 1981), hot-film or hot-wire anemometry (Hasenkam et al., 1987, 1988), pulsed Doppler ultrasonic velocimetry (Farahifar et al., 1985), and a variety of flow visualization techniques (Affeld et al., 1989; Knoch et al., 1988). However, although a lot of experimental studies have been carried out, a detailed and complete description about the flow field of a BS valve is still not available, especially in the vicinity of the valve disc. Up to now, the most detailed velocity measurements around the valve were conducted by Yoganathan and his colleagues (Yoganathan et al., 1978, 1979), but they measured only the axial-velocity component; no 3-D information about the flow field is available. Hasenkam et al. (1987, 1988) and Hanle et al. (1989) have used 3-D data presentation techniques and obtained cross-sectional velocity distributions in the aorta downstream. No results in the vicinity of the valve are available. Figliola and Mueller (1981) have measured the velocity in the vicinity of the valve only in the plane of symmetry. Khalighi et al. (1983) and Chandran et al. (1985) have studied the flow development in a model aorta with different orientations of the BS valve and measured the transverse flow in the ascending aorta. They found a birehtical secondary flow distal to the valve. Farahifar et al. (1985) have done similar studies.
with an elastic aortic arch model. Tiederman et al. (1986) have used two-component LDA in the measurement of velocity profiles in the vicinity and downstream of the valve and found circumferential flows. However, no further results and explanation were given. The flow visualization of Affeld et al. (1989) and Knoch et al. (1988) showed only qualitative results.

Numerical calculation of the flow field is expected to overcome most of the physical limitations on experimental measurements and to yield the velocity distribution of the entire flow field. At present, both the finite-difference method (FDM) and the finite-element method (FEM) have been used in the study of valve flow fields. Mueller (1978) presented an excellent review on the application of FDM to physiological flows. However, because of the special complex geometry of the tilting disc valve, only a few studies about its flow field were carried out numerically. Greenfield and Au (1976) calculated the flow field of a tilting disc valve by means of FDM at low Reynolds number; Peskin and McQueen (1980) numerically simulated the flow field of a BS valve in mitral position; and Idelsohn et al. (1985) have studied the flow field of a BS valve in aortic position by using the FEM with maximum Reynolds numbers up to 1500. However, all of these numerical calculations were carried out on the basis of 2-D models. The question arises whether these results can be used to simulate the real flow field. Theoretically, a non-axisymmetric flow field cannot be calculated by means of a 2-D model. However, because of the difficulties of conducting 3-D calculations, 2-D calculations are often used to estimate 3-D flow fields. But the feasibility needs to be verified by comparison with experimental results. Unfortunately, most of the present experimental studies were concentrated on the turbulent flow field under high Reynolds numbers and there is a lack of experimental data at lower Reynolds numbers corresponding to numerical calculations.

In this paper, first a theoretical appraisal about the flow field around a BS valve is given. Then the results are reported from detailed experimental measurements using LDA, carried out for verification in the vicinity and downstream of the valve under steady flow conditions with Reynolds numbers from 210 to 800. Finally, results from numerical calculations using FEM based on a 2-D model are presented.

THEORETICAL MODEL

The disc of a BS valve is just a circular plate. Approximately, the flow field of this valve is similar to the flow field around a circular plate in an infinite flow. Due to the complicated mathematical analyses involved, it is nearly impossible at present to give a quantitative theoretical description about that kind of flow field. For this reason, here only a qualitative description about the features of this flow field is given as an illustration.

Figure 1 shows the flow around a circular plate with different incidence angles in an infinite flow field (Thwaites, 1960). When the disc is set perpendicular to the stream like the caged disc valve, the flow separates along the whole circumference and a large axisymmetric wake is formed behind the disc [Fig. 1(a)]. However, if the disc is set at a certain angle with respect to the approaching flow, the wake is no longer

Fig. 1. The flow past a circular plate: (a) normal to a uniform stream; (b) at a large incidence; (c) at a small incidence, showing the formation of the spiral vortex flow. (From Thwaites, 1960.)
offset axisymmetric and is distorted into the pattern sketched in Fig. 1(b). Features to note particularly are the upward flow induced along the sides of the wake, and the tendency for a downward flow along the top of the wake. When the incidence angle of the circular plate becomes smaller, these two features are emphasized still more and the wake downstream rapidly forms itself into two cores, around each of which the fluid flows in a spiralling fashion [Fig. 1(c)]. The BS valve corresponds to the case of a smaller incidence angle at its fully open position, and the curved sinus boundary will enhance the transverse flow; so, very likely, there exist two spiral vortex streams downstream of the valve. The experimental results of Affeld et al. (1989), Chandran et al. (1985) and Tiederman et al. (1986) show some features of the existence of this phenomenon. Further evidence about the existence of spiral vortex flows will be given in the next section.

EXPERIMENT

Experimental procedure and apparatus

Figure 2 shows the experimental set-up and the coordinate system for LDA measurement, as well as the axial measuring sections. The experimental set-up consists of a test section, a reservoir, a steady flow pump (UERDER 2032) and connecting tubes. The test section consists of an annular sinus, a downstream aorta and a conical inlet with an enlarged short channel before it, simulating the left ventricle. The sizes of the inlet and the sinus are taken approximately from the measuremental results of Reul et al. (1990) and Swanson and Clark (1974). Because a 16 mm Björk–Shiley standard prosthesis is used, the inner diameter of the aorta and the valve ring are made equal to 16 mm and all other dimensions are based on this size. The test section as well as the upstream and downstream connecting tubes are made of perspex due to the use of LDA. To avoid optical distortion of the curved tube wall, the whole test section is contained within a square box with parallel flat planes outside (also made of perspex). The BS valve is mounted with its disc plane perpendicular to the x–y plane and keeps fully open during the velocity measurements [Fig. 2(b)]. It is remarked here that the orientation of the valve disc is different from that in Fig. 1.

To get velocity profiles, a one-component forward-scattering reference-beam laser Doppler anemometer (DISA) was used with a measuring volume of 200 μm in length and 60 μm in width. The flow rate was obtained by using an electromagnetic blood flow meter (Transflow 601). Because of the multiple curvatures of the sinus wall, a mixture of oil and kerosine was used as the fluid which exactly matches the refractive index of the perspex aortic model and allows performance of LDA velocity measurements in the vicinity of the valve. Due to the high kinematic viscosity of the oil mixture, the maximum Reynolds number could reach only 850 according to the maximum flow rate of the pump. This value is comparable to the maximum value used in the numerical calculations. To get more information at higher Reynolds numbers, water was used, but only the velocity profiles at upstream and downstream aorta sections could be obtained. Here, the Reynolds number is defined as $Re = UD/v$, in which $U$ is the cross-section-averaged velocity in the aorta, $D$ is the inner diameter of the valve ring, and $v = μ/ρ$ is the kinematic viscosity of the fluid. In addition, the viscosity of the oil mixture varies with temperature (Rindt et al., 1990); so whenever the oil mixture was used, it was heated to and kept at 40°C to reduce its viscosity to about 0.1164 cm² s⁻¹. Since the electromagnetic flow meter cannot be used in combination with the oil mixture because of its non-ionicity, the Reynolds number was estimated on the basis of the upstream velocity profile, which is parabolic when $Re \leq 800$.

Data acquisition and processing

Data acquisition was performed by a measuring system consisting of an IBM personal computer, a 4-channel signal processing unit and an interface. The voltage signal coming from the tracker was fed to the signal processing unit. There it was adjusted between $-5 \text{ V}$ and $+5 \text{ V}$ and finally transferred into velocity data by using an AD-converter. Before making graphs, the velocity at each point was averaged over 120 to 180 samples depending upon the Reynolds number. The sampling frequency used was 300 Hz.

In this study, the velocity profiles at different $y$ and $z$ positions of the seven cross-sections indicated in Fig. 2(b) were measured at Reynolds numbers 210, 450 and 800, respectively, when an oil mixture was used. When water was used, the Reynolds number reached 4000 but only the velocity profiles at upstream and aorta sections in the diametric plane ($y = 0$ or $z = 0$) could be measured due to the optical distortion induced by the
curved tube wall together with the non-matchable refractive indices of water and perspex. The velocities at the wall and valve surfaces were presumed to be zero according to the no-slip boundary condition. Usually, the velocity measurement was made from wall to wall with non-standard steps. But when the measurement was taken very near the disc surface [Fig. 6(a) and (b)], only one half of the flow field \((z \leq 0)\) could be measured due to the obstruction of the valve to the laser beams.

The transverse flow in the \(y\) direction was measured by rotating the plane spanned by the two laser beams by 90°. The positive flow direction was defined as the positive \(y\) direction. Unfortunately, the velocity component in the \(z\) direction could not be obtained. To get this velocity component, a backward-scattering LDA should be used.

**Results**

Figure 3 shows the axial-velocity profiles in the plane \(z = 0\) at \(Re = 800\). The most obvious feature of the flow field of a tilting disc valve is the separation of the flow duct into two unequal orifices, the major orifice and the minor orifice. The approaching flow has a parabolic velocity profile at the upstream section. When the flow goes through the valve section, it forms two nearly equal-intensity jet flows; then, as it moves downstream, it skews towards the major orifice side. This figure also shows recirculating flows in the sinus but none in the aorta nor under the valve. The variation of the velocity profiles with the Reynolds number at sections V2, S2 and A1 is shown in Fig. 4. It can be seen from Fig. 4(c) that the downstream skewness of the velocity profiles and the absence of recirculation in the aorta remain unchanged even at \(Re = 4000\).

The axial-velocity profiles at \(Re = 800\) in the planes of constant \(y\) value are shown in Fig. 5. The difference between the velocity profiles in different planes is very obvious. Figure 5(a) shows nearly uniform or flat velocity profiles in the flow duct of the major orifice, while Fig. 5(c) shows a narrow jet flow in the minor orifice. Three-peak velocity profiles at the valve and sinus (VS) sections and two-peak velocity profiles in the aorta are very clear in Fig. 5(b) and (c), which correspond to the minor-orifice flow duct. The central

![Fig. 3. Axial-velocity profiles in the plane \(z = 0\) at \(Re = 800\), showing the separation of flow duct into major and minor orifices and the skewing of the flow in downstream aorta.](image)

![Fig. 4. Axial-velocity profiles in the plane \(z = 0\) at sections V2 (a), S2 (b) and A1 (c) for \(Re = 210 (~\times~), 450 (~\triangle~), 800 (~\square~), 1000 (~\oplus~), 1500 (~\circ~), 2400 (~\land~), 4000 (~\blacksquare~). Note that the velocity scale in (c) is different because water was used when \(Re > 800\).](image)

![Fig. 5. Axial-velocity profiles at \(Re = 800\) in the planes of constant \(y\) value are shown in Fig. 5. The difference between the velocity profiles in different planes is very obvious.](image)
Analyses of the steady flow field around a RS valve

Fig. 5. Axial-velocity profiles in the planes of (a) \( y = -5 \text{ mm} \), major-orifice flow duct; (b) \( y = 0 \), central plane (V1: \( y = -0.75 \text{ mm} \); V2: \( y = 1 \text{ mm} \)) and (c) \( y = -4 \text{ mm} \), minor-orifice flow duct, at \( Re = 800 \).

peak of the three-peak velocity profiles grows higher towards the middle (around \( y = -4 \text{ mm} \)) of the minor orifice [Fig. 5(c)]. The variation of the velocity profiles with the Reynolds number at different \( x \) positions in the plane \( y = 0 \) or just under the valve, is shown in Fig. 6. Figure 6(a)–(d) shows the three-peak velocity profiles at the VS sections, and Fig. 6(e) shows the two-peak velocity profiles at aorta section A1. The higher the Reynolds number, the more evident this phenomenon becomes. (Theoretically, these profiles should be symmetric, but in reality this condition is difficult to satisfy.) From this figure, we can also see that there is a low but non-zero velocity area under the valve, and a small reverse flow near the valve’s trailing edge at \( Re = 800 \) [Fig. 6(a) and (b)]. This low-velocity area is also visible in Fig. 7, where the axial velocity profiles in the entire valve section V1 are shown.

Figure 8 shows the transverse flow in the \( y \) direction, measured in planes of constant \( y \) value, and Fig. 9 shows the variation of the transverse flow with the Reynolds number. An upward flow (towards the major orifice) in the central area and a downward flow (towards the minor orifice) near the wall are very clear from these figures. Such a flow phenomenon exists only in the minor-orifice flow duct. There is more downward flow than upward flow near the major-orifice flow duct [Fig. 9(a) and (d)]. In the plane \( y = -4 \text{ mm} \), they become nearly equal [Fig. 9(c)]. A maximum transverse velocity of about 37 cm s\(^{-1}\), which is nearly half of the maximum axial velocity [about 86 cm s\(^{-1}\)], was found at section S2 [Fig. 9(d)]. According to the features of spiral vortex flow (Fig. 1), these results show clearly the existence of two spiral vortices downstream of the valve, each of which occupies half of the flow field with its centre located about 1/4 diameter from the wall [Fig. 9(b)].

**NUMERICAL CALCULATION**

**Physical model**

For numerical studies, a 2-D model was used. The Björk–Shiley standard prosthesis at its fully open position (60° opening angle) was used as the prototype (Fig. 10). The geometry of the model aorta is the same as that of the \( z = 0 \) plane of the experimental model (Fig. 2b). The studies were performed under the following assumptions: incompressible Newtonian isothermal fluid, steady laminar flow, fixed valve, rigid walls and negligible body forces.

**Governing equations and finite-element formulation**

The incompressible fluid flow described above must satisfy the Navier–Stokes equation and the continuity equation in the following non-dimensional form:

\[
\frac{1}{Re} \nabla^2 u + (u \cdot \nabla) u + \frac{1}{\rho} \nabla p = f, \\
\nabla \cdot u = 0,
\]

where \( Re \) is the Reynolds number, \( u \) the velocity vector with components \((u, v)\), \( p \) the pressure, \( f \) the boundary force and \( \nabla \) the gradient vector operator. The boundary conditions used for the above problem are:

- no-slip condition at the wall: \( u = v = 0 \),
- inlet condition: \( u = u_0 (1 - r^2/R^2) \), \( v = 0 \),
- stress-free outlet condition: \( u_n = v_n = 0 \).

With equations (1) – (3), the velocity and pressure fields can be completely determined.

For numerical simulation of the flow field, the finite-element method was used. By applying Galerkin’s method, a set of finite-element equations can be obtained:

\[
S u + N(u) u + \tau I M^{-1} L u = f,
\]

which contains only velocity unknowns due to the employment of a penalty function method. Here \( u \) is the velocity vector, \( S \) the diffusion matrix, \( N(u) \) the convection matrix, \( I \) the divergence matrix, \( M \) the pressure matrix and \( f \) the right hand-side vector, and \( \tau = 1/\epsilon \), where \( \epsilon \) is a very small penalty factor of the order of \( 10^{-6} \). For more details about this method and the deriving procedure, the reader is referred to Cuvelier et al. (1986) and Horsten (1990).

**Solution procedure**

The finite-element mesh used in this study is illustrated in Fig. 10(a), with refinement near the leading and trailing edges of the valve disc. Due to the existence of a non-linear convective term, linearization is necessary. Two linearization methods, the Picard method and Newton method (Cuvelier et al., 1986),
were used. The solution of the system of FEM equations was arrived at by means of iterative methods. Usually, one Picard iteration was applied before the Newton iteration because the Picard method converges linearly with a larger radius of convergence.

Fig. 6. Axial-velocity profiles at sections V1 [(a), $y = -0.75$ mm], V2 [(b), $y = 1$ mm], S1 (c), S2 (d) and A1 (e) in the $y=0$ plane for $Re=210 (-\times -)$, 450 ($-\triangle -$), 800 ($-\square -$), 1000 ($-\ast -$), 1500 ($-\diamond -$), 2400 ($-\rightarrow -$), 4000 ($-\blacksquare -$). [In (a) and (b) only half of the flow field could be measured due to obstruction of the laser beams by the valve. Note the different velocity scales used.]

Fig. 7. Axial-velocity profiles at valve section V1. $Re=800$.

Fig. 8. Transverse-velocity profiles ($y$ direction) in the planes of (a) $y=0$ and (b) $y=-4$ mm at $Re=800$. 
Analyses of the steady flow field around a BS valve

Fig. 9. Transverse-velocity profiles (y direction) at section S1 in the planes of y = 3.4 mm (a), y = 0 (b) and y = -4 mm (c), and at sections S2 (d) and A1 (e) in the plane y = 0 for Re = 210 (- x -), 450 (- △ -), and 800 (- □ -). The existence of two vortices is indicated in (b).

while Newton’s method converges quadratically but with a smaller radius of convergence. The iteration process started at a lower Reynolds number. Then Re was increased step by step by reducing the kinematic viscosity. The solution of Stoke’s equation [equation (4) without the convective term] was used as an initial guess of the solution at the very beginning of the iteration process. After that, the preceding solution of the Navier-Stokes equation at a lower Re was used as the initial value for the next iteration process. Using this technique, the maximum Reynolds number \( \text{Re}_{\text{max}} \) reached 700. A higher \( \text{Re}_{\text{max}} \) could have been reached if the mesh were refined or a smaller increment of Re were taken. But this would cost much more computer time. The computations were performed on an Apollo minicomputer by means of the SEPRAN software package (Segal, 1988). Both velocity and pressure fields as well as shear stress distributions were calculated, but only the results for the velocity field are discussed in this paper.

Results

The results of the numerical calculations are shown in Figs 10 and 11. Figure 10(b)–(e) shows the velocity vectors at different Reynolds numbers. At smaller Reynolds numbers (< 200), there is only a small vortex in the lower sinus (minor orifice side). As the Reynolds number increases, a central vortex in the minor-orifice outflow region appears and grows rapidly, which forces the minor orifice jet to move towards the sinus. There is no recirculation area in the downstream aorta near the lower aortic wall (corresponding to the minor orifice) at all Reynolds numbers. However, when the Reynolds number exceeds 400, a recirculation area near the upper aortic wall occurs and grows up very quickly. Figure 11 shows the velocity profiles at different positions. Compared with Fig. 4, those profiles show similar features as the experimental result, especially when \( \text{Re} \leq 262.5 \). When Re is larger, the minor-orifice jet flow becomes much stronger and narrower. The major difference between the numerical
and the experimental results are the backflows in the central area and near the upper aortic wall.

**DISCUSSION AND CONCLUSION**

In this study, a large amount of experimental data of the flow field around a BS valve have been obtained, especially the detailed measurement of transverse-velocity profiles near the valve, which was not reported in literature earlier. The feature of upward flow in the central area and the downward flow near the walls is evident on the transverse-velocity profiles taken both at sinus sections and at aorta sections (Figs 8 and 9), which reveals the existence of two spiral vortex flow streams downstream of the valve. According to the features of the spiral vortex flow, the three-peak velocity profiles at the VS sections [Fig. 6(a)-(d)] reinforce this conclusion. These peaks are actually produced by the central minor-orifice jet and the two spiral vortices. The two-peak axial-velocity profiles in the aorta [Fig. 6(e)] can be explained as follows: when the flow moves downstream, the jet flow is absorbed by the two spiral vortices; so the central peak on the three-peak velocity profile disappears. Khalighi et al. (1983) also obtained the transverse-velocity profile and the three-peak axial-velocity profile at a section near the sinus edge, and found the biehelic flow in the ascending aorta which could not be introduced by the aortic arch. Tiederman et al. (1986) found strong circumferential flows just downstream the valve disc. These facts further proved the existence of spiral vortex flow. In fact, the major orifice has a large inlet and a relatively smaller outlet. The fluid flow in the major orifice will find its way into the minor orifice through side orifices between the valve disc and the sinus wall, which forms two downward flows along the wall. (The near-wall downward flows can also be formed due to the reduced outlet of the sinus.) The two downward flows meet in the centre of the minor orifice, and then turn upward. Finally, two spiral vortices are formed.

The flow field of a BS valve is separated into two jet flows by the valve disc, the major-orifice jet and the minor-orifice jet. This is a well-known feature of its flow field. From Figs 3 and 4, it can be seen that the maximum velocities of the two jets are nearly equal at the valve sections. However, the minor-orifice jet is very narrow, while in comparison the major-orifice jet is much wider (Figs 5 and 7). This should be the reason for the downstream skewness of the flow towards the major-orifice side. Furthermore, the spiral vortex flow should also have an influence on the skewness of the velocity profiles because of its relationship with the multi-peak velocity profiles. From Figs 4(c) and 6(e), it can be seen that the maximum velocities of the two-peak velocity profiles in the $y=0$ plane are nearly the same as those of the corresponding skewed velocity profiles in the $z=0$ plane. That means the skewness of velocity profiles occurs only in a central plane around $z=0$, and not in a parallel plane of $z=0$ near the wall.

The experimental results of this study, the multi-peak velocity profiles in the $y=0$ plane and the skewed velocity profiles in the $z=0$ plane [Figs 4(c) and 6(e)], are in agreement with the corresponding experimental results of other investigators (Hasenkam et al., 1987, 1988; Yoganathan et al., 1979; Khalighi et al., 1983; Chandran et al., 1985; Farahifar et al., 1985). The experimental results of Figliola and Mueller (1977, 1981) are different from this study, showing a long recirculation area along the aortic wall corresponding to the minor orifice. The reason perhaps is because they used an enlarged aorta section in their model. In addition, Yoganathan et al. (1978) observed a stagnation area in the minor-orifice outflow region at high Reynolds numbers. Our experiment shows that there is no obvious stagnation area in the same region when $Re<800$. But the measurement at section V2 shows that there is a small reverse flow near the valve's trailing edge at $Re=800$ [Fig. 6(b)]. It seems possible
Analyses of the steady flow field around a BS valve

that stagnation areas first occur near the valve edges and then develop into the centre as the Reynolds number increases.

The 2-D numerical calculation in this study can simulate the skewed velocity distribution in the aorta when \( Re < 400 \) [Fig 10(b)-(c)]. However, the numerical result of a large recirculation area near the aortic wall corresponding to the major orifice does not agree with the experimental results. The vortex flow in the central area would be possible at a higher Reynolds number according to Yoganathan's results (1978), but it is not realistic at lower Reynolds numbers (\( Re \leq 700 \)) compared with the experimental results in this study (Fig. 3). From Fig. 3, we can see that the minor-orifice jet flow is separated from the valve disc and nearly parallel to the aortic wall. It seems that the numerical calculation could simulate this feature. However, it is obviously impossible to simulate the 3-D behaviour of the BS valve flow field by using such a 2-D model calculation. Idelsohn et al. (1985) have obtained numerically the flow field of a model BS valve at a higher Reynolds number (1500). But their results, which show a large recirculation area in the aorta, occupying nearly half of the flow duct, also do not agree with the experimental results of this study.

In short, from this study it can be concluded that:

1. The steady flow field around an aortic Björk–Shiley standard tilting disc valve is completely 3-D, featured by two spiral vortex streams and by two orifice-jet flows.
2. There is no recirculation zone in the downstream aorta.
3. Due to the extremely non-uniform spatial distribution of the velocity, the numerical calculation based on 2-D models could not exactly simulate the non-axisymmetric flow field of a BS valve. To obtain detailed and complete information about this flow field, 3-D calculations are suggested.

In this study, detailed experimental measurements of the flow field around a BS standard valve in the vicinity and downstream of the valve have been carried out, especially under lower Reynolds numbers, which is very useful for the verification of 3-D numerical models. In our opinion it is worthwhile starting the 3-D calculation under steady flow conditions with a model similar to our experimental model, so that the results are easy to compare with the experimental results of this study.

In addition, our velocity measurement in the vicinity of the valve was conducted only under steady flow conditions with \( Re \leq 800 \), but the velocity measurement in the aorta shows that the main features of the velocity profiles remain unchanged as \( Re \) goes up to 4000 [Figs 4(c) and 6(e)]. This is supported by similar results of other investigators who conducted their studies under steady flow conditions with high Reynolds numbers (Khalighi et al., 1983; Hasenkam et al., 1987, Yoganathan et al., 1979) or under physiological pulsatile flow conditions (Chandran et al., 1985, Tiederman et al., 1986). So, the flow pattern shown in this study seems very likely to exist at higher Reynolds numbers or during some forward-flow phases in physiological pulsatile flows. To verify this, further studies under high Reynolds numbers and physiological pulsatile flow conditions are necessary.

Acknowledgements—This research is supported by the Dutch Technology Foundation (STW) (grant no. FWT 85-0857) and
an EEC fellowship from the Liaison Committee of Rectors Conferences of Member States of the European Communities, and partly supported by the National Natural Science Foundation of China. The authors thank Dr P. H. M. Bovendeerd for his valuable suggestions in the preparation of this paper.

REFERENCES


