FLOW ESTIMATION OF A DOUBLE OUTPUT CENTRIFUGAL ARTIFICIAL HEART PUMP AS A BIVENTRICULAR ASSIST DEVICE BY COMPUTATIONAL FLUID DYNAMICS

DC. Sin[A,B], A.CC. Tan[A,B] , YJ. Kim[C], N. Kelson[D] and G. Chadwick[B]
[A] Institute of Health and Biomedical Innovation, Queensland University of Technology, Brisbane, Australia.
[B] Faculty of Built Environment and Engineering, Queensland University of Technology, Brisbane, Australia.
[C] Faculty of Health, Queensland University of Technology, Brisbane, Australia.
[D] Division of Technology, Information and Learning Support, Queensland University of Technology, Brisbane, Australia.

Abstract

Some of the mechanical assisted devices include pneumatic pumps, axial flow pumps, and centrifugal pumps. Among the pumps available, the centrifugal pumps have attracted worldwide interest because centrifugal flow pumps have high efficiency. However, the motion of centrifugal pump impeller relative to the housing to achieve the required physiological outputs has led to many challenges in its fluidic design, flow dynamics from impeller blades, output profile of the volute and very small clearances between the impeller and the housing. The computational fluid dynamics (CFD) package FLUENT was applied to simulate the flows through the Centrifugal blood pump. In this study the three-dimensional flow patterns through the artificial heart pump were predicted. The predicted velocity profiles and volume flow rates reflect possible blood stagnation, which may contribute to thromboembolism.

Introduction

Due to the shortage of donor hearts for transplant, the need and demand for artificial heart pumps has been well documented (Chua et al. 2005). This shortage has lead to worldwide investigation and development of mechanical assisted devices such as the Ventricular Assist Device (VAD). Some of the mechanical assisted devices include pneumatic pumps, axial flow pumps, rotary pumps and centrifugal pumps (Wood et al. 2005).

Among the pumps available the centrifugal has attracted worldwide interest for use as a VAD(Mussivand, Hasle et al. 2004; Wu, Allaire et al. 2004), because they permit compact design and can achieve high delivery rates. The QUT artificial heart pump project team has already designed a prototype continuous flow VAD based on a centrifugal pump, for use in adults, which is approximately 50mm in diameter. In addition, the team has constructed a mock circulation loop and a 4:1 scaled-up test rig for the purpose of conducting experimental flow visualization studies of the prototype.

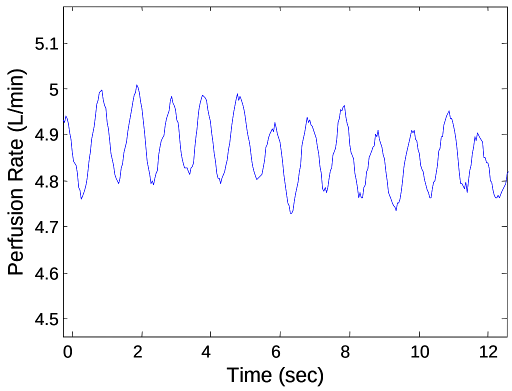

Figure 1 - Transition of perfusion with VAD Support.

The mock circulation loop(Timms, Tan et al. 2005) was developed to imitate the needed features of circulatory systems. This rig provides the ability to assist a failing heart to maintain sufficient cardiac output (Fig.1) and incorporates pulsatile left and right circulatory systems.

The flow behaviour in the narrow clearance of VADs can not be analysed through experimental technique because the flow rate is not constant during systolic and diastolic periods. The flow field occurs in a small area and constructing the entire flow field requires a lot of time. However, a computational Fluid dynamics (CFD) model provides an effective method in order to investigate the geometric parameters of the newly developed pump and possible thrombus formation sites.

In the present work, a Computational Fluid Dynamics (CFD) model of the 4:1 scaled-up prototype test pump (Fig.2) has been developed. CFD simulation is a powerful tool for design, demonstration and optimization of small pumps intended for use as a VAD or BiVentricular Assist Device (BVAD) because it offers a convenient and efficient means for the analysis of flow patterns. The CFD flow predictions can also be used to predict thrombosis formation and haemolysis, which are two types of blood damage that represent major limiting factors in the use of centrifugal pumps in real applications involving VADs or total heart replacement.

At this stage, the CFD results were compared with flow visualisation testing at non-pusatile condition at diastolic period in order to establish the validity and reliability of the computational model. In this study the three-dimensional flow patterns through the artificial heart pump were predicted. The predicted velocity profiles and volume flow rates reflect possible blood stagnation, which may contribute to thromboembolism.

Materials and Methods

Nondimensional scaling approach

In a study of rotary blood pumps, Throckmorton A.L et al [3] illustrated how to scale test data from one pump impeller to another of differing size for the same pump configuration. Many texts also discuss similarity laws, which support this approach (Yamane et al. 2004). Table 1 shows the design parameters and operating conditions for the designed pump and scaled-up pump using similarity laws. For example, in a pump working with an incompressible fluid, the volume flow rate (*Q*) and the total pressure rise (ρgH) are combined into two dimensionless variables to characterise the pump performance:

Head coefficient: $$\Psi = gH\Omega^{-2}D^{-2} \quad (1)$$

Flow coefficient: $$\varphi = Q\Omega D^{-3} \quad (2)$$

Where Ψ represents the Head coefficient, φ is the flow coefficient, D signifies the impeller's diameter, Ω indicates the rotational speed (*rev/min*), and Q denotes the flow rate in LPM. The trendline from pressure coefficient versus the flow coefficient yields a mathematical relationship which may be used as a reasonable estimation for scaling up the pump.

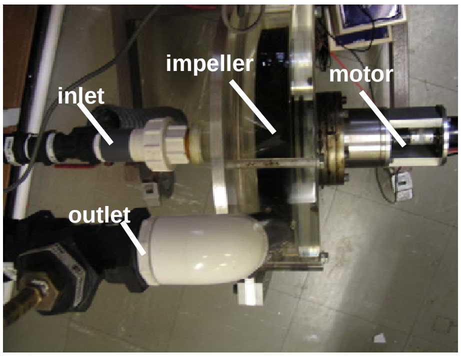

Figure 2 - Photo of QUT Scaled-up Blood Pump.

Table 1. The design parameters and operating conditions for the prototype and scale-up pump

Charateristic	**Prototype**	**Scale-up**
Impeller diameter	0.05 m	0.2 m
Number of balde	6	6
Design rotor speed	3000 rev/min	60 rev/min
Design flow rate	5 l/min	6.4 l/min
Design pressure rise	13000 Pa	83 Pa
Viscosity	3.5e-3 Pa s	1.003e-3 Pa s
Density	056 kg/ m^3	1000 kg/ m^3

CFD Analysis

In this study, the fluid flowing in the pump was assumed to be Newtonian with a viscosity of 1.003e-3 *Pa s* and a density of 1000 kg/m^3 (Tsukamoto, Ito et al. 2001; Song, Wood et al. 2004). The boundary conditions were set according to the operating conditions of the pump: the flow rate inlet was set as 0.0239 kg/s, and the pressure outlet was 1500 *Pa* at 125 *rev/min*. The flow was modelled as steady with a standard k-ε model for turbulence.

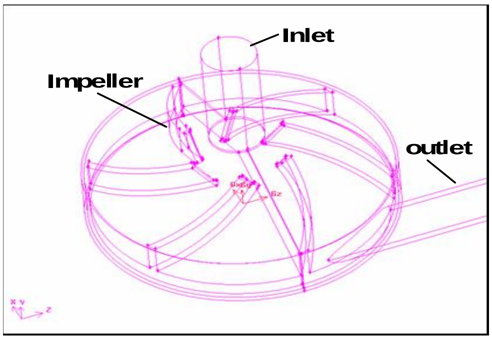

Figure 3 - 3D view of the simulation model of AHP.

To develop the computational model, the finite-volume CFD code FLUENT 6 (Fluent Inc.) was employed. The 4:1 scaled-up model included the flow region of the entire centrifugal pump, and a three-dimensional view of the whole model and an impeller with meshes are illustrated in Fig. 3. Blood enters the pump through the straight inlet pipe and turns 90˚ in the radial direction at the stationary cap, then passes through the impeller passages to the volute and subsequently flows out through the pump exit. There are two gap regions in the model, one lying between the upper impeller shroud and upper pump housing, and other one between the lower impeller shroud and lower pump housing. Both regions require careful modelling if reliable simulation results are to be obtained.

Results and Discussion

These streamline and velocity profiles can be examined along any plane of interest in the computational fluid field to identify regions of recirculation, stagnation and large gradients resulting in fluid stresses (Okada, Masuzawa et al. 1999). The Highlighted box in Fig. 4 shows a region of the volute tongue with the exit diffuser, and indicates possible undesirable flow patterns. Recirculation is usually accompanied by stagnation and large energy losses (Anderson, Wood et al. 2000; Throckmorton, Untaroiu et al. 2004), therefore, it should be eliminated to reduce the possibility of blood damage and improve the fluid dynamic efficiency.

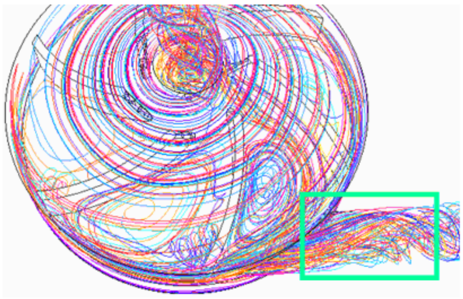

Figure 4 - Flow pattern at off design condition.

Subsequent to the numerical analysis, flow visualisation using a high speed camera of the test pump was carried out to validate the predicted results in Fig. 5, and also to aid in assessing the pump performance. The pressure symmetry is destroyed at diastolic condition, because volute flow must either accelerate or decelerate to exit the fixed throat area. Stagnation also occurs at the water-cult, because the impeller discharge angle and volute angle are no longer matched(Timms, Hayne et al. 2005; Timms, Tan et al. 2005).

Figure 5 - Flow visualisation test around water-cut.

Conclusions

The pump is designed for a double output centrifugal artificial heart pump for use as a Bi-Ventricular Assist Device (Bi-VAD). A non-dimensional scaling technique based on existing blood pump design was explored and utilised for scaling up the prototype pump for the flow visualization purposes. The scaled-up AHP has been modelled using CFD with a flow rate of 0.0239*kg/s* and impeller speed of 125*rev/min*. The flow patterns from the analysis indicate that recirculation occurs at the outlet. This suggests that the scaled-up pump should be modified in the diffuser region to eliminate the recirculation and reduce the possibility of blood damage and improve the fluid dynamic efficiency. Although CFD can evaluate any physical quantity at any position the results highlight the limitations of CFD and the need for careful construction of meshes and selection of suitable mathematical models and correct boundary conditions.

References

Anderson, J. B., H. G. Wood, et al. (2000). Computational Flow Study of the Continuous Flow Ventricular Assist Device, Prototype Number 3 Blood Pump. *Artificial Organs* **24**(5): 377-385.

Mussivand, T., D. A. Hasle, et al. (2004). Left ventricular assist system: clinical freedom from device replacement. *The Journal of Heart and Lung Transplantation* **23**(2, Supplement 1): S147.

Okada, Y., T. Masuzawa, et al. (1999). Design of Centrifugal Type Artificial Blood Pump Using Combined Motor Bearing. *Preceedings of DETC'99* - ASME Design Enginerring Techinical Conferences, Las Vegas.

Song, X., H. G. Wood, et al. (2004). Computational Fluid Dynamics (CFD) study of the 4th generation prototype of a continuous flow Ventricular Assist Device (VAD). *Journal Of Biomechanical Engineering* **126**(2): 180-187.

Throckmorton, A. L., A. Untaroiu, et al. (2004). Computational Analysis of an Axial Flow Pediatric Ventricular Assist Device. *Artificial Organs* **28**(10): 881-891.

Timms, D. L., M. Hayne, et al. (2005). A Complete Mock Circulation Loop for the Evaluation of Left- Right- and Bi- Ventricular Assist Devices. *Artificial Organs* **29**(7): 564-572.

Timms, D. L., A. C. C. Tan, et al. (2005). LVAD Pump Performance And Force Characteristics In A Pulsatile Complete Mock Circulation Loop. *Artificial Organs* **29**(7): 573-580.

Tsukamoto, Y., K. Ito, et al. (2001). Computational Fluid Dynamics analysis for centrifugal blood pumps. *Journal of Congestive Heart Failure and Circulatory Support* **1**(4): 337-343.

Wu, Y., P. E. Allaire, et al. (2004). A Bridge from Short-term to Long-term Left Ventricular Assist Device : Experimental Verification of a Physiological Controller. *Artificial Organs* **28**(10): 927-932.

Wood G. H, Throckmorton L. A., Untaroiu A., Song X. (2005). The medical physics of ventricular assist devices. *Reports on Progress in Physics* **68**:545-576.