Mecánica Computacional Vol. XXIII, pp. 1483-1489
G.Buscaglia, E.Dari, O.Zamonsky (Eds.)
Bariloche, Argentina, November 2004

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF A NOVEL AXIAL FLOW BLOOD PUMP WITH TWO COUNTER-ROTATING IMPELLERS

[*]Martín Chimenti, [*]Lucas Varela, [‡]Eduardo de Forteza, [‡]Roberto Favaloro

[*]Industrias Metalúrgicas Pescarmona SA (IMPSA). Mendoza. Argentina

[‡] Fundación Favaloro. Buenos Aires. Argentina

Key words: Axial flow blood pump, Counter-rotating impellers, Computational Fluid, Dynamics Analysis, Axial flow.

Abstract. *Computational Fluid Dynamics (CFD) has been used for developing and evaluating a novel axial flow blood pump. The pump embodiment includes two counter-rotating impellers with independent motorization. This device could be used as an implantable pump for boosting blood circulation in patients whose hearts are not pumping at an adequate cardiac output. The blood pump was designed for a flow rate of 5 L/min and 100 mmHg of head pressure and a combination of counter-rotational speeds of 8400 rpm and 3900 rpm for the inlet and outlet impellers respectively. From the CFD results it is concluded that an embodiment including two counter-rotating impellers is hydraulically feasible.*

INTRODUCTION

The design of rotary blood pumps is a trade-off among several features. Antitraumatic and antithrombogenic characteristics in this type of cardiac assist devices are difficult to obtain simultaneously (1).

Axial pumps generally include an impeller and a stationary diffuser with fixed blades at the outlet of the impeller (2-4). A stationary diffuser de-accelerates and redirects the flow to boost pump performance. However, it can generate stagnation and recirculation zones where probability of undesirable blood clot formation is increased (5-8). The novel design under study has a second counter-rotating impeller instead of a stationary diffuser. The flow at the pump outlet is axial (absolute velocity vectors parallel to the rotation axis) for predetermined combinations of counter-rotational speeds of the impellers.

Computational Fluid Dynamics (CFD), which is widely used currently as a tool to analyze and test rotary blood pumps (9), has been used for design, development and validation of the hydraulic concept of two counter-rotating impellers.

Materials and Methods

The novel design includes a pump housing and two counter-rotating impellers. Each impeller has three blades and an 8 mm diameter hub. The fluid flows in a 15 mm diameter duct (Figure 1).

If we assume one-dimensional, stationary and ideal incompressible flow, then the continuous fluid flow behavior through a rotary pump fitted with blades is mathematically defined by the Euler equation (10-12).

According to the Euler equation an axial flow is obtained for a given speed of the outlet rotor, moreover if an axial inflow is considered, the following expression is concluded (13):

$$\frac{g.H_2}{\eta_2.\omega_2} = -\frac{g.H_1}{\eta_1.\omega_1}$$

ω_i, η_i, H_i , g angular velocity, efficiency, head of each impeller and acceleration of gravity.

Where sub-indices 1 and 2 are for the first and second rotor respectively.

Having (H_1 and H_2) and (η_1 and η_2) positive signs, the previous equation is valid only if ω_1 and ω_2 have opposite signs. This is the reason why the proposed configuration includes two counter-rotating impellers. A stationary diffuser is no longer necessary in this arrangement.

Counter-rotational speeds of 8400 rpm and 3900 rpm for the inlet and outlet impellers respectively were set for this analysis

Bench Test

H-Q curves of the pump were obtained in a bench test using a solution of glycerin and water with a viscosity of 3 cp. The pump was connected to a 500 mL reservoir with ½ in. PVC tubes, and a screw clamp was applied to produce the require conditions. Flow (L/min) and differential pressure (mm Hg) were measured for fixed combinations of counter-rotating speeds

Numerical method and CFD model

A finite volume numerical solution technique and multiple frames of reference were used. ICEM CFD Hexa (AEA Technology, Ontario, Canada) was used for grid generation. CFX-TASCflow (AEA Technology) CFD software package was used for boundary conditions setup, solving and post-processing of results.
Runs with increasing mesh refinement were performed to determine sensitivity of the computed solution to the mesh size. Two models of turbulence were analyzed, a standard k-ε model with a mesh of 137,747 nodes and a k-ω model with a mesh of 271,679 nodes. The principal difference between these two meshes is the finer near wall mesh for the k-ω model.
An axial flow of 5 L/min was set as a boundary condition at the inlet. Given that the flow was considered incompressible and we are interested in differential values, an arbitrary pressure of zero Pascal was set at the outlet. Frozen-rotor interface at the sliding face was used for simultaneous simulation of both counter-rotating impellers.
A skew of 3 and a physical advection correction were used for discretization. All walls were considered hydraulically smooth. Blood was assumed to be a Newtonian fluid with a density of 1.055 Kg/m^3, and a viscosity of 3 cp

K-ε analysis

As a first step of a general efficiency analysis a standard k-ε method was utilized, which is widely used for turbulence models (14-20). It has proven to be stable and numerically robust. A logarithmic velocity profile and a scalable wall function were set as a wall boundary condition. The wall-function approach in CFX-TASCflow is an extension of the method of Launder and Spalding (21). The near wall tangential velocity is related to the wall-shear-stress by means of a logarithmic relation. *Scalable* wall-function formulation developed by CFX can be applied on arbitrarily fine grids and allows the user to perform a consistent grid refinement independent of the Reynolds number of the application. The basic idea behind the scalable wall-function approach is to assume that the surface coincides with the edge of the viscous sublayer, which is defined to be at Y+ = 11. This is the intersection between the logarithmic and the linear near wall profile. The computed Y+

is not allowed to fall below this limit. Therefore, all grid points are outside the viscous sublayer and all fine grid inconsistencies are avoided.
The time step was reduced to 0.01 (s) to promote convergence.

K-ω analysis

k-ω SST (Shear Stress Transport Model) model of turbulence is more adequate for low Reynolds number. It was used for a more detailed study of the fluid flow. A fine enough grid was used near the walls in order to have approximately a Y+ number below two in all walls.
One of the major problems of standard two-equation turbulence models is that they often fail to predict the onset and the amount of flow separation under adverse pressure gradient conditions. The k-ω based Shear-Stress-Transport (SST) model of Menter (22) was designed to give a highly accurate representation of the separation phenomena by the inclusion of transport effects into the formulation of the eddy-viscosity. The superior performance of this model has been demonstrated in a large number of validation studies (Bardina et al).
The time step was set to 0.0002 (s) in this case to promote convergence.

RESULTS AND DISCUSSION

Figure 3 shows experimental and CFD characteristics curves of a pump with two counter rotating impellers.
Four combinations of rotational speeds were evaluated experimentally. These curves show the range of operation of the pump and its capacity for working in different regimens. One combination of rotational speeds was evaluated numerically using a k-ε and k-ω models of turbulence. The k-ε model agree well with the experimental data at values of flow rates close to 5.5 L/min and the k-ω model agree better for values of flow rates close to 5 L/min. However for greater values of flow rate experimental differential pressure values are below CFD analysis and the opposite for lower values of flow rate. A similar tendency was reported by Wood and Mitoh (23-24), This disagreement has not been solved yet and further study is required to solve this problem.

CONCLUSIONS

According to results, a two counter-rotating impellers pump could be a new approach for the solution of the left ventricle assist problem. Additional experimental in-vitro tests are necessary for design validation. CFD is a useful tool for hydraulic design and development, however it is a one-sided perspective.
Correlation of results with experimental data must be improved and further studies are required.

FIGURES

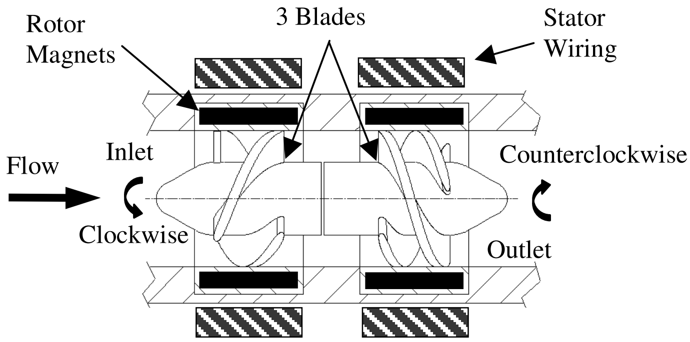

Figure 1. Pump configuration

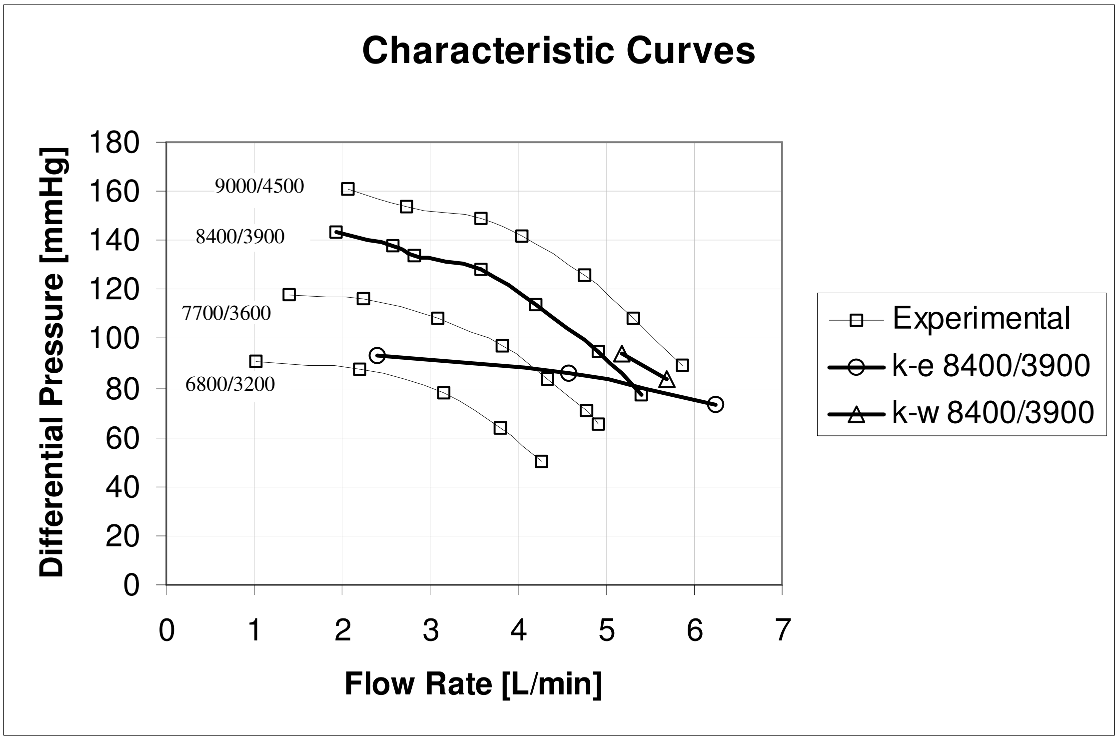

Figure 2.

REFERENCES

1. Nose Y. Design and Development Strategy for the Rotary Blood Pump. Artif Organs 1998;22:438-446.
2. Butler KC, Dow JJ, Litwak P, et al. Development of the Nimbus/University of Pittsburgh Innovative Ventricular Assist System. Ann Thorac Surg. 1999;68:790-794.
3. Damm G, Mizuguchi K, Bozeman R, et al. In Vitro Performance of the Baylor/NASA Axial Flow Pump. Artif Organs 1993;17:609-613.
4. Macris MP, Myers TJ, Jarvik R, et al. In Vivo Evaluation of an Intraventricuolar Electric Axial Flow Pump for Left Ventricular Assitance.ASAIO J 1994;40:M719-M722.
5. Jarvik R. United States Patent US 5,824,070. October 20, 1998.
6. Burgreen GW Antaki JF, Griffith BP. A design improvement strategy for axial blood pumps using computational fluid dynamics. ASAIO J 1996;42:M354-M360.
7. Thomas DC, Butler KC, Taylor LP et al. Continued development of the Nimbus/University of Pittsburgh (UOP) axial flow left ventricular assist system. ASAIO J 1997;43:M564-M566
8. Burgreen GW, Antaki JF, Wu J et al. A computational and experimental comparison of two outlet stators for the Nimbus LVAD. ASAIO J 1999;45:328-333
9. Burgreen GW, Antaki JF, Wu ZJ, Holmes AJ. Computational fluid dynamics as a development tool for rotary blood pumps. Artif Organs 2001;25:336-340.
10. White FM Fluids Mechanics. Mc Graw Hill, third edition.
11. Shepherd DG. Principles of Turbomachinery, Macmillan, NY, 1956
12. Dixon SL. Fluid Mechanics and Thermodynamics of Turbomachinery (3erd edition). Pergamon Press, Oxford, 1981
13. Kazatchkov L, Varela L. United States Patent US 6,247,892 B1. June 19, 2001.
14. Masuzawa T, Tsukiya T, Endo Seiko et al. Development of design methods for a centrifugal blood pump with a fluid dynamics approach: results in hemolysis test. Artif Organs 1999;23:757-761
15. Miyazoe Y, Sawairi T, Ito K et al. Computational fluid dynamics analysis to establish the design process of a centrifugal blood pump: second report. Artif Organs 1999;23:762-768
16. Tsukamoto Y, Ito K, Sawairi T et al. Computational fluid dynamics analysis of a centrifugal blood pump with washout holes. Artif Organs 2000;24:648-652
17. Watanabe N, Karsak O, Neudel F et al. Simulation of the BP-80 blood pump. Artif Organs 2001;25:733-739.
18. Takano T, Schulte-Eisturp S, Kawahito S et al. Inlet port positioning for a miniaturized centrifugal blood pump. Artif Organs 2002;26:45-48.
19. Sartori-Montecroce O, Goubergrits L and Affeld K. Investigation of the flow performance of a nutating blood pumo by computational fluid dynamics. Artif Organs 2002;26:392-396

20. Chan WK, Wong YW, Ding Y et al. Numerical investigation of the effect of blade geometry on blood trauma in a centrifugal blood pump. Artif Organs 2002;26:785-793.
21. Launder BE. and Spalding D. B. The numerical computation of turbulent flows. *Comp Meth Appl Mech Eng*, 3:269-289, 1974
22. . Menter. FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA-Journal.*, 32(8), 1994.
23. Wood HG, Anderson J, Allaire PE, McDaniel JC, Bearnson G. Numerical solution for blood flow in a centrifugal ventricular assist device. Int J Artif Organs 1999;22(12):827-36.
24. Mitoh A, Yano T, Sekine K et al. Computational fluid dynamics analysis of an intra-cardiac axial flow pump. Artif Organs 2003;27(1):34-40.