Investigating the Use of Turbulence Models for Flow Investigations in a Positive Displacement Ventricular Assist Device

Mohammed G. Al-Azawy, Ali Turan, and Alistair Revell

School of Mechanical, Aerospace and Civil Engineering, The University of Manchester, Manchester, England, UK

Abstract— **Computational fluid dynamics (CFD) is applied to study the hemodynamics of flow inside a pulsatile pump left ventricular assist device (LVAD), in order to evaluate the hemolysis and thrombus formation. The positive displacement or pulsatile pump, which includes valves and a pusher plate (to mimic the natural heart), is the focus of this study. Turbulence is observed to play an important role in the accuracy of predicted levels of shear stress and strain rate, both of which are crucial in assessing the long term feasibility of the device.**

In order to obtain this aim, three turbulence models have been used: a standard Reynolds stress model (RSM), the shear stress transport (SST) *k*-omega, and Transition-SST, in addition to laminar flow i.e. "no model". An unstructured mesh was created for the simulation and the motion of the pusher plate was created via a dynamic mesh layering method. Valves were simulated in their full open position, to mimic the natural scenario. Results were allowed to reach a periodic state and were validated with available experimental data. The results indicated that the RSM gave the best agreement with the experimental data.

Keywords— **Ventricular assist device, pulsatile pump, Dynamic mesh, computational fluid dynamics, Turbulent flow.**

I. INTRODUCTION

In recent years, artificial heart devices have emerged as a promising alternative therapy for patients suffering from heart disease. They are particularly attractive given that the number of available donor hearts (hearts for transplant) is very small, and in general far lower than potential demand.

A ventricular assist device (VAD) is a kind of artificial heart device that is used to support the heart's pumping function for either short or long term. The main pumping chambers of the natural heart are the ventricles, and the majority of pumping is undertaken by the left ventricle.

In general, the main issues to be taken into account in the hemodynamics of heart pumps are thrombus and hemolysis; which are directly related to flow inside the pump. The blood flow through the natural ventricles, arteries, heart assist devices or artificial heart pumps might exhibit laminar, turbulent, or transitional flow. Indeed, the physiological Reynolds numbers could range from the laminar region to peak Reynolds numbers of the order of 10,000 in mechanical assist devices. However, in positive displacement pumps, the levels of turbulence would be higher due to flow separation from the flow from the inlet port around the valves and into the chamber. The periodic nature of the flow would induce a cyclic transition from beat to beat (valve occluder), producing a complex flow inside the VAD chamber [1]. Previous studies have shown that the sudden expansion of the flow from the inlet to the main chamber of a LVAD can cause transient to turbulence at a Reynolds number as low as 754 [2].

Avrahami assumed that the flow inside the Berlin pulsatile VAD was laminar based on a mean Reynolds number of 1350, whereas the peak Reynolds number reached 4200 and the flow might have become transitional during the deceleration of the diastole. Moreover, Avrahami mentioned that the assumption of laminar flow might introduce inaccuracies, particularly with regards to the valve downstream [3]. Jason [4] performed an experimental work using PIV on the new designs (V-2, V-3, and V-4) of the 50cc Penn State LVAD. The study focused on changing the position and orientation of the outlet port and investigated the general influence of this change on the flow field within the chamber and, more specifically, on thrombus formation.

The positive displacement pump which mimics the natural heart is the focus in this study. The aim of this study is to investigate the ability of various turbulence models to predict the flow inside the left ventricular assist device (50cc Penn State LVAD design V2), and to evaluate the impact of the predictive uncertainties on problems of blood damage which have been closely related to the fluid dynamics within ventricular assist devices.

II. PHYSICAL MODEL

The model to be investigated is the 50cc Penn State Left Ventricular Assist Device (LVAD) (V2 design). Figure 1a shows the V2 design, which illustrates the position of Bjork-Sheily valves and the pusher plate. The mitral valve (23 mm) and aortic valve (21 mm) were simulated without supported struts for the sake of computational simplicity.

The model was investigated under physiological operating conditions at 86 BPM (beats per minute) and 4.2 LPM (liters per minute). According to the movement of the pusher plate, the resultant flow rates are shown in Figure 2, with

I. Lacković and D. Vasić (eds.), *6th European Conference of the International Federation for Medical and Biological Engineering*,
IFMBE Proceedings 45, DOI: 10.1007/978-3-319-11128-5_99

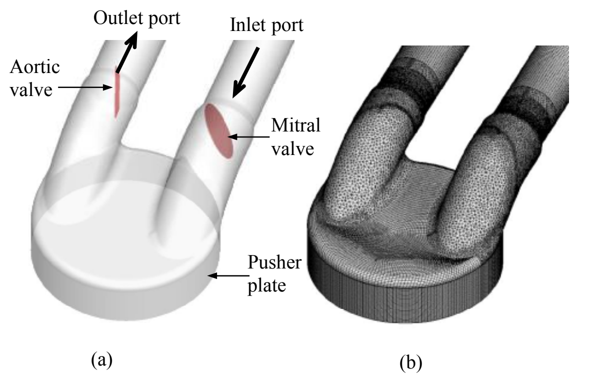

Fig. 1 The model geometry showing, a) fully opened mitral valve in 30^0 orientation and fully opened aortic valve in a 0^0 orientation at inlet and outlet ports respectively, b) Unstructured mesh used for simulation.

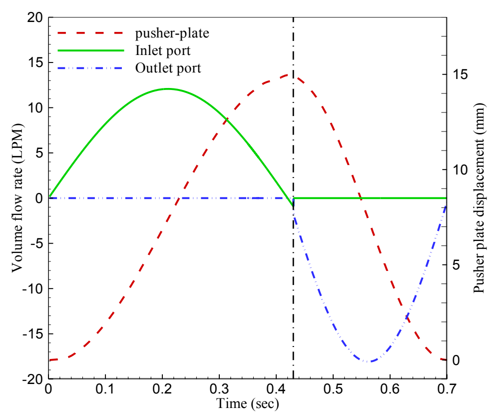

Fig. 2 Flow rates of inlet and outlet ports and pusher plate movement; the vertical dashdot line illustrates the onset of systolic (aortic valve opens) and end of diastolic (mitral valve closes).

a peak systole flow rate of 18 LPM and a peak diastole flow rate of 12 LPM.

III. Computational details

Five different mesh models were created to investigate the spatial mesh resolution for the three-dimensional simulations (number of cells at onset of diastole M1:755 060, M2:1 375 566, M3:1 965 151, M4:2 313 005, and M5:2 785 928), as shown in Figure 1b. Figure 3 shows the variation of x-velocity with mesh size. These meshes were compared at the end of the diastolic phase. Mesh M4 is adequate to capture the properties of the flow within the chamber and near the valves and is selected for the following sections. A prism mesh was used to resolve the boundary layer of the moving pusher plate, with the first mesh point near the wall located at the non-dimensional distance $y^+ < 1$ inside the chamber and < 2.4 in the entire device.

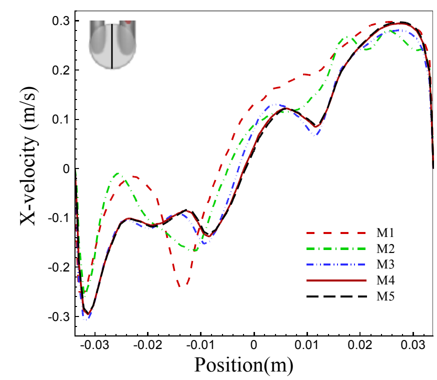

Fig. 3 X-velocity at vertical line (8mm from the front face).

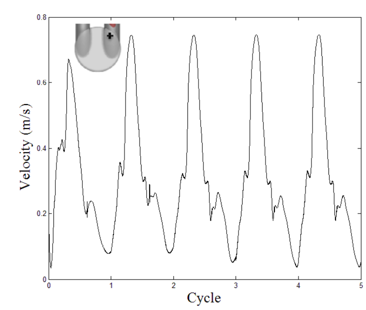

Fig. 4 History of velocity magnitude for five cycles at one points of flow.

The sensitivity to time step was investigated and a value of $\triangle t = 1 \times 10^{-3}$ was found to be satisfactory, resulting a maximum Courant-Friedrichs-Lewy (CFL) number of around 1 inside the chamber. The simulation was allowed to continue until a time periodic flow was obtained. Figure 4 shows the history of velocity magnitude at one point in the chamber and the test performed for five pump cycles of flow with the same condition. In the current study, the fourth cycle has been chosen to extract the data of the simulation.

IV. Modelling of the valves and pusher plate movement

In the current study, the mitral and aortic valves were fixed in the fully open position during the pump cycle. To mimic the closed valve an interface was fixed immediately above the valve, which was set to be a wall during one part of the cycle and an open interface during the next. During the diastolic phase, the flow enters from the inlet port as the pusher plate expands, and the interface above the aortic valve will be set as a wall; whereas during the systolic phase the pusher plate will pump the flow towards the outlet port and instead, the interface above the mitral valve will be defined as a wall. The same procedure will then be repeated for other cycles.

The motion of the pusher plate was modelled using a dynamic mesh layering method, which acts to add/remove cells with a height of 0.5 mm. The time of the diastolic phase is longer than the systolic, with the velocity of the wall introduced as follows:

For the diastolic phase:

$$V_{wall} = A * \frac{2\pi}{T} * sin(\frac{2\pi}{T} * t) \quad (1)$$

For the systolic phase:

$$V_{wall} = A * \frac{2\pi}{T} * cos(\frac{2\pi}{T} * t) \quad (2)$$

Where t is the flow time (sec), A is the distance between the moving wall and the mid-stroke position, and T is a one cycle period.

V. Computational simulation and turbulent models

All the simulations implemented unsteady computational flow for a full pumping cycle of the three dimensional analysis. The simulations were solved via a finite volume code (ANSYS FLUENT V.14) in order to solve the conservation of mass and Navier-Stokes equations and assumed an incompressible, Newtonian fluid. The pressure-velocity coupling is obtained by using the SIMPLEC algorithm, and discretization for the pressure and the velocities are based on PPRESTO and a second order upwind scheme respectively. For temporal discretization a first-order implicit scheme is applied.

In the simulation the total pressure and static pressure were set at the inflow and outflow respectively according to *in vitro* measurements. Therefore, the total pressure was set to zero at the inlet and the static pressure was set to 80 mm Hg at the outlet, in order to achieve the 80 mm Hg average pressure rise.

Turbulence plays an important role in the prediction of strain rate and shear stress. These are important factors in the evaluation of blood damage, which may manifest in forms such as thrombus and hemolysis. CFD researchers of blood flow have assumed that most of the blood flow in the cardiovascular system is laminar (Re is usually 300 and sometimes less)[1].

From the results, the peak mitral Reynolds number was recorded as 3000. For the sake of accuracy, the flow should be compared with a turbulent model. In the current study, three models of turbulent flow (shear stress transport (SST) *k*-omega [5], Transition-SST [6], and Reynolds-stress model (RSM)[7]) were investigated, as well as ”laminar” model in which no model is used.

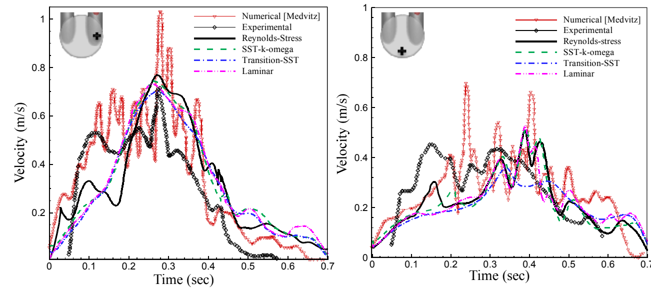

Fig. 5 Velocity magnitude comparison at 3 o’clock (left) and 6 o’clock (right) points.

VI. Results and discussion

Results were validated by comparing the instantaneous flow field in the device against the available PIV experimental data. The comparisons mainly encompassed the instantaneous velocity magnitude at extraction points in the chamber on a plane 3 mm from the front face of the chamber. The details of experiments in a mock circulatory loop were illustrated by Hochareon [8]. *In vitro* PIV and numerical data published by Medvitz [9] were used in the computational comparisons.

Figure 5 shows the comparison of velocity magnitude at two points inside the chamber, these points were located 25.725 mm from the center of the chamber at 3 mm from the front face, using the models of turbulent and laminar flow against the experimental work and the numerical study. From the results of the comparison, a reasonable validation with the experimental work was obtained,particularly with regard to the results of Reynolds stress model (RSM);in addition, the results of the SST-k omega test were more acceptable than those derived from the Transition-SST model and the laminar test. In order to analyze the behavior of flow inside the blood pumps, the shear stress and strain rate can be incorporated with blood damage models related to the shear stress and exposure time.

Figure 6 illustrates the two dimensional strain rate histories. The strain rate which calculated in equation 3 useful to identify the areas of flow stagnation or stasis.

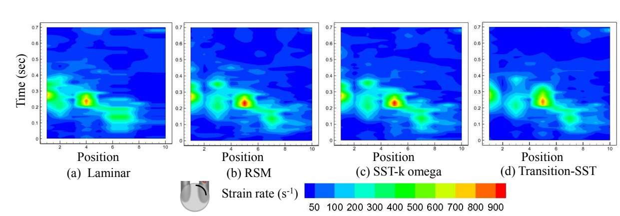

Fig. 6 Strain rate history in arc 12 o’clock to 3 o’clock within the chamber (2.3 mm from the wall).

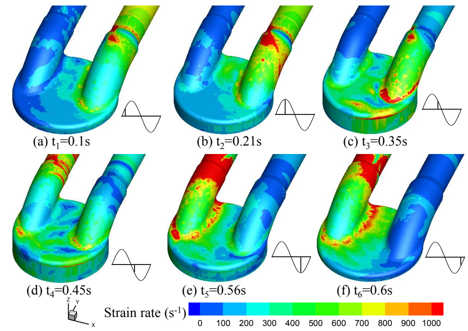

Fig. 7 Wall strain rate at early, peak, and late diastolic and systolic phase.

The strain rate tensor calculated as:

$$S_{ij} = \frac{1}{2}(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) \tag{3}$$

The following results are from the RSM model only. Figure 7 illustrates the strain rate observed at the walls of the device in early, peak, and late diastolic and systolic phase. The authors observed that the maximum shear rate were near the mitral valve in peak diastole and near the aortic valve in peak systole. From the investigation of flow inside the chamber, the area near the top of the chamber is more prone to thrombosis since the strain rate less than 500 s^{-1}, in the majority of time, as shown in figure 8. It is also observed that the strain rate near to the valves is always higher than that inside the chamber, due to the high velocity near the valve therefore we need in future to investigate the shear stress to know the level of the hemolysis.

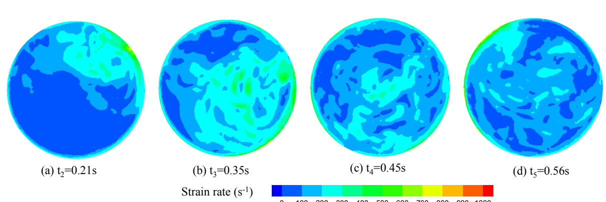

Fig. 8 Strain rate at peak, late diastole and early, peak systole .

VII. CONCLUSION

Three turbulence models (RSM, SST-k omega, and Transition-SST) as well no model have been employed in the unsteady simulation at positive displacement pump LVAD with moving the pusher plate by layering method and full open valves. The comparisons with available experimental and numerical data indicate best agreement for a RSM model.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the assistance given by IT Services and the use of the Computational Shared Facility at The University of Manchester.

CONFLICT OF INTEREST

The authors declare that they have no conflict of interest.

REFERENCES

1. Lee Waite Jerry Fine. *Applied Biofluid Mechanics*. McGraw Hill 2007.
2. Konig C S Clark C, R Mokhtarzadeh-Dehghan M. Investigation of unsteady flow in a model of a ventricular assist device by numerical modelling and comparison with experiment. *Medical engineering & physics.* 1999;21:53–64.
3. Idit Avrahami, Moshe Rosenfeld, Shmuel Einav. The hemodynamics of the Berlin pulsatile VAD and the role of its MHV configuration. *Annals of biomedical engineering.* 2006;34:1373–88.
4. Nanna Jason C, Wivholm Jennifer A, Deutsch Steven, Manning Keefe B. Flow field study comparing design iterations of a 50 cc left ventricular assist device. *ASAIO journal(American Society for Artificial Internal Organs).* 2012;57:349–357.
5. Menter F. R.. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal.* 1994;32:1598-1605.
6. Menter F. R., Langtry R. B., Likki S. R., Suzen Y. B., Huang P. G., Volker S.. A Correlation-Based Transition Model Using Local Variables Part I: Model Formulation. *Journal of Turbomachinery.* 2006;128:413-422.
7. B. E. L. G. J. Reece, Rodi W.. Progress in the Development of a Reynolds-Stress Turbulence Closure. *J. Fluid Mech..* 1975;68:537-566.
8. Hochareon Pramote. *Development of particle image velocimetry (PIV) for wall shear stress estimation within a 50cc Penn State artificial heart ventricular chamber.* PhD thesisThe Pennsylvania State Universiy 2003.
9. Medvitz Richard B, Reddy Varun, Deutsch Steve, Manning Keefe B, Paterson Eric G. Validation of a CFD methodology for positive displacement LVAD analysis using PIV data. *Journal of biomechanical engineering.* 2009;131:111009 1-9.

Author: Mohammed G. Al-Azawy
Institute: School of MACE, The University of Manchester
Street: Princess street
City: Manchester
Country: United Kingdom
Email: mohammed.al-azawy@postgrad.manchester.ac.uk