An Overset Mesh Approach for Valve Closure: An LVAD Application

Mohammed G. Al-Azawy[1,2], A. Turan[1] and A. Revell[1]

[1]*School of Mechanical, Aerospace and Civil Engineering, The University of Manchester, Manchester, U.K.*

[2]*Mechanical Engineering Department, College of Engineering, Wasit University, Wasit, Iraq*

Keywords: Computational Fluid Dynamics (Cfd), Left Ventricular Assist Device (Lvad), Overset Mesh, Non-Newtonian Flow, Turbulence Modelling.

Abstract: A comprehensive Computational Fluid Dynamics (CFD) simulation of transient, non-Newtonian, and turbulent blood flow through a positive displacement pump, left ventricular assist device (LVAD), is executed. Non-Newtonian blood flow is conducted to investigate the flow through a pulsatile pump LVAD by using common blood viscosity model: Carreau. The numerical results of non-Newtonian fluid with a turbulence model, Elliptic Blending Reynolds Stress Model (EB-RSM) are presented. The computational domain that has been selected is a pulsatile pump, which includes valves and a moving pusher plate. An overset mesh zero gap technique was employed to capture the cyclic motion of pusher plate and valves rotation to mimic the scenario of a natural heart. The use of this technique to rotate the valves and ensure full valve closure presented a good agreement results with the experimental data.

1 INTRODUCTION

Fluid mechanical studies inside artificial heart pumps have been ongoing since the 1970s in an attempt to investigate and understand the flow behaviour of blood inside the device in order to predict and mitigate the blood damage caused by the device.

Artificial heart valves have been used widely in order to assist or replace the natural damage valves. Previously, the researchers observed that serious problems associated with the flow around the valves such as separated and secondary flow, high pressure and large turbulence shear stress (Kiris et al., 1997; Stijnen et al., 2004). The main causal risks associated with heart devices, especially with the valves, are thrombosis and haemolysis, both of which are directly related to the flow field within the devices.

In simulating the blood flow inside the cardiovascular systems, one of the important issues that needs to be taken into account is how to treat the nature of blood flow as a fluid in order to accurately predict and evaluate the wall shear stress and strain rate. Local haemodynamics are not only affected by the geometry of artificial heart assist devices, properties of flow as pulsation or not, but also by the natural properties of blood. Therefore, in the current study, a non-Newtonian Carreau model has been used to investigate the blood flow within a left ventricular assist device (LVAD).

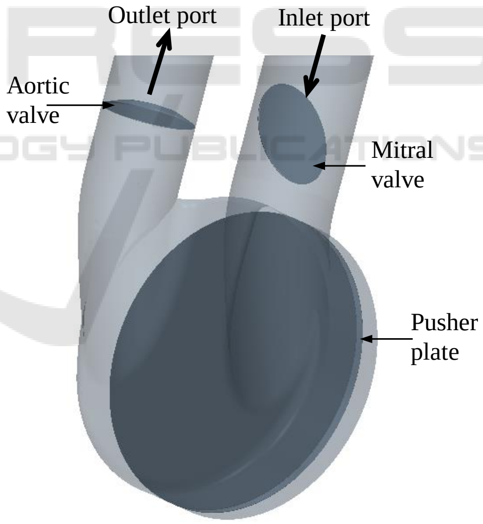

Figure 1: Model Geometry.

A study in 2004 by Yin et al. (Yin et al., 2004) was performed experimental and numerical study to measure *in vitro* the procoagulant properties of platelets induced by flow through Carbomedics bileaflet and Bjork-Shiley monoleaflet mechanical heart valves in a left ventricular assist device. A modified prothrombinase method was used to measure the platelet activation states during circulation. Wilcox $k-\omega$ turbulence model and platelet shear-stress histories were used to simulate the CFD model of turbulent, transient

Al-Azawy, M., Turan, A. and Revell, A.
An Overset Mesh Approach for Valve Closure: An LVAD Application.
DOI: 10.5220/0005663901450151
In *Proceedings of the 9th International Joint Conference on Biomedical Engineering Systems and Technologies (BIOSTEC 2016) - Volume 1: BIODEVICES*, pages 145-151
ISBN: 978-989-758-170-0

and non-Newtonian blood flow. The results from the platelet activation states measurements showed that the bileaflet MHV activated platelets at a rate more than twice that observed with the monoleaflet MHV.

For simplicity, most numerical simulations assumed the flow around the heart valves to be two dimensional (Bluestein et al., 2000) flow and fixed fully open position such as (Medvitz et al., 2009). The techniques that can be used for valve simulation is discussed in this work. Kreider et al. (Kreider et al., 2006) conducted an experimental work (planer particle image velocimetry) to analysis the flow field associated with the Bjork-Shiley mechanical heart valve of the 50cc Penn State ventricular assist device. The authors noticed that there was longer duration wall washing motion occurring at 45 degree. In most recent study by the authors (Al-Azawy et al., 2016; Al-Azawy et al., 2015), the valves were assumed in the fully open position. The authors investigated unsteady flow inside a 50cc LVAD Penn State, design V2. The authors tested the laminar and turbulent flow to assess the sensitivity to a range of commonly used turbulence models. In this study, six turbulence models have been used: shear stress transport (SST) $k-\omega$, transition SST, Spalart-Allmaras, $k-\varepsilon$, Reynolds stress model (RSM), and laminar model. The CFD model includes valves and a moving pusher plate, the valves were simulated in their fully open position and a layering method was employed to move the pusher plate and capture the cyclic motion. The results were validated with the numerical and experimental data and showed that the RSM provided the best agreement with the experimental data over much of the flow. In parallel with this work, the authors in this study extended the investigation to use the overset mesh technique to rotate the valves and pusher plate movement. The overset mesh zero gap approach has been employed to incorporate full valve opening/closing, instead of assuming full opening position as illustrated in the previous work. The aim of this study is to evaluate the impact of the overset mesh approach on the flow around the valves.

2 NUMERICAL MODELLING

In the current study, a model of a ventricular assist device is constructed. The model to be investigated is a 50cc LVAD test rig V2 design, as described in previous studies (Medvitz, 2008; Al-Azawy et al., 2016). This design includes Bjork-Sheily valves and pusher plate. The valves were simulated without supported struts for the sake of simplicity; see Figure 1, which shows the mitral valve (23 mm) in its fully open position (70 deg) and the aortic valve (21 mm) in its fully closed position (0 deg). The model was examined under physiological operating conditions at 4.2 LPM (litres per minute) and 86 BPM (beats per minute).

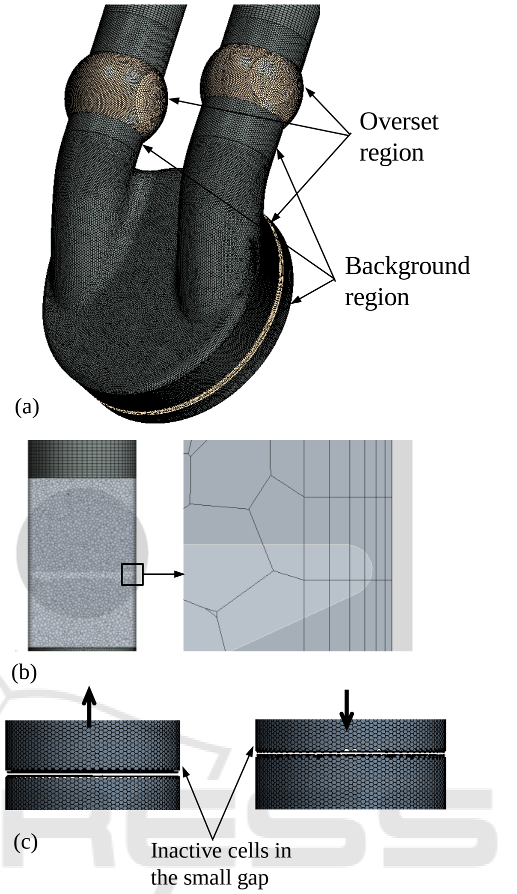

Figure 2: Snapshots of (a) mesh configuration showing the overset and background regions, (b) the gap between the background and valve showing the cells and (c) the space after initialisation.

All the simulations employed unsteady computational flow for a full pumping cycle of the three-dimensional analysis. The simulations were implemented using a finite volume code STAR-CCM+ 10.02, a commercially available CFD package (Star-CCM, 2015), to solve the Navier-Stokes equations:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu(|S|) + \mu_t)\frac{\partial u_i}{\partial x_j}\right] \tag{2}$$

where u_i is the velocity in i direction ($i = 1,2, and 3$), $u_i = (u,v,w)$, correspond to the coordinate system, $x_i = (x,y,z)$, respectively; p is the pressure; and ρ is the density. In the current study we

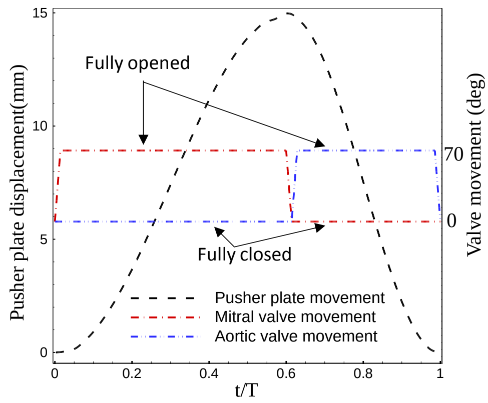

Figure 3: Valves and pusher plate movement.

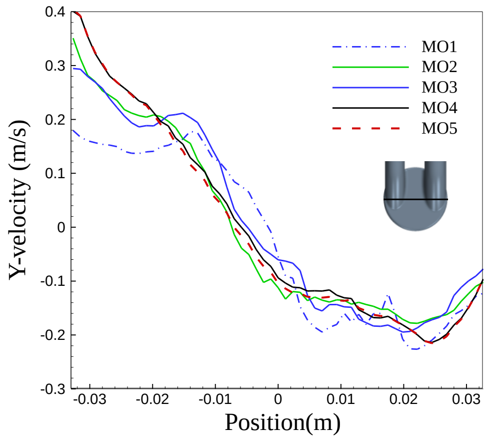

Figure 4: Time-averaged y-velocity located on the plane $z/z_c = 0.53$ along a horizontal centreline at time $t/T = 0.614$.

have used the non-Newtonian Carreau models, $\mu(|S|)$ is the blood viscosity that depends on the shear rate magnitude,$|S| = \sqrt{2S_{ij}S_{ij}}$ where $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$. The turbulent viscosity, μ_t, calculated via additional transport equations representative of the turbulence.

The following expression has been given for the Carreau model (Carreau, 1972; Johnston et al., 2004):

$$\mu(|S|) = \mu_\infty + (\mu_0 - \mu_\infty)(1 + (\lambda S)^2)^{(n-1)/2} \tag{3}$$

where λ is the relaxation time constant ($\lambda = 3.313s$), $n = 0.3568$, μ_0 is the viscosity of blood at zero shear rate ($\mu_0 = 0.056pa.s$) and μ_∞ is the infinite shear viscosity ($\mu_\infty = 0.00345pa.s$).

The transport equations are solved by using a segregated flow approach and the SIMPLE algorithm for pressure-velocity coupling. A second order implicit unsteady scheme is applied in time, while the spatial discretisation utilises a second-order upwind scheme along with a hybrid Gauss-LSQ method is used for gradient reconstruction. In the present work, the total and static pressures were set according to *in vitro* measurements which indicated a device mean static pressure rise of 80 mm Hg (Medvitz, 2008). Therefore, the total and static pressures were set to zero at the inlet and 80 mm Hg at the outlet respectively.

3 DESCRIPTION OF THE VALVES AND PUSHER PLATE MOVEMENT

In positive displacement pump problems, in order to acquire the desired scenario for the diastolic and systolic phases, it is necessary to model the valve closure and the pusher plate movement to maintain unidirectional flow. The times of the valve closing and opening are short compared to the duration of diastole and systole phase. Therefore, in previous study by (Al-Azawy et al., 2016) the mitral and aortic valves were fixed in the fully open position during the pump cycle and to mimic the closed valve as in a natural heart. Further to this, an interface was fixed above the valves and introducing it as a wall on one occasion and as an open interface on another, depending on the cycle phase. However, there are various methods to model the valves; either a dynamic mesh, immersed boundary (Peskin, 2002) or a binary flow model (where the flow is either fully closed or fully open). A variable viscosity model was implemented by Medvitz (Medvitz et al., 2007) to model the valve closing, as used by Avrahami (Avrahami, 2003) and Stijnen (Stijnen, 2004). In the present study the authors extended the investigation and used the overset mesh technique to model the valve rotation, as the authors believe this will give more accurate results and more insight analysis for shear stress and strain rate near the valves.

The overset mesh technique (Chimera) has been successfully used to simulate the flow among the moving regions with zero gap between the moving and stationary zones. The overset mesh zero gap interface method has been employed to simulate the pusher plate movement inside the chamber and valve rotation with full closure of the valve, as shown in Figure 2a, which illustrates the background and overset regions. In the background zone, where the active cells are found, the regular discretised governing equations are solved, while in the overset mesh, which contains the inactive or passive cells (these cells will be active during the movement) no equation is solved. The inactive cells that separate active and passive are called the acceptor cells and the active cells along the interface with the inactive cells are called donor cells. The interpolation is used to express variable values at acceptor cells via variable values at donor cells (Star-

CCM, 2015).

The closing and opening time of the valves was short, taking approximately 10 ms (Medvitz, 2008). Figure 3 illustrates the movements of the pusher plate and valves; during the diastole the flow enters after opening the mitral valve as the pusher plate expands to fill chamber. The mitral valve then closes at the end of diastolic, whereas during the systole the aortic valve will open and the pusher plate will pump the flow towards the outlet port and the aortic valve will close at the end of the systolic period. The cycle is then repeated. The thickness of the chamber was varied cyclically from a minimum of $z/z_c = 0.21$ to a maximum of $z/z_c = 1$, where z was the distance from the front face of the chamber and the chamber thickness z_c was 18.8mm corresponding to a maximum volume of approximately 50cc.

The time of the diastolic phase is ($0 \leq t < 0.43$s) while the time of the systolic phase is ($0.43 \leq t < 0.7$s), with the velocity of the wall introduced as follows:

For the diastolic phase:

$$V_{wall}^{diastole} = A\frac{2\pi}{T}sin\left(\frac{2\pi}{T}t\right) \tag{4}$$

For the systolic phase:

$$V_{wall}^{systole} = A\frac{2\pi}{T}cos\left(\frac{2\pi}{T}t\right) \tag{5}$$

where V_{wall} is the velocity of the moving wall, the pusher plate is represented as a function of time, 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 ($T = 0.7s$).

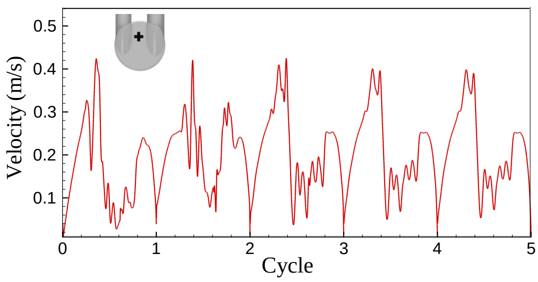

Figure 5: Evolution of velocity magnitude at point in the chamber.

4 COMPUTATIONAL DETAILS

The following five computational meshes have been used: MO1 (957,573), MO2 (1,511,511), MO3 (2,944,787), MO4 (4,030,158), and MO5 (5,614,830). These meshes were created by

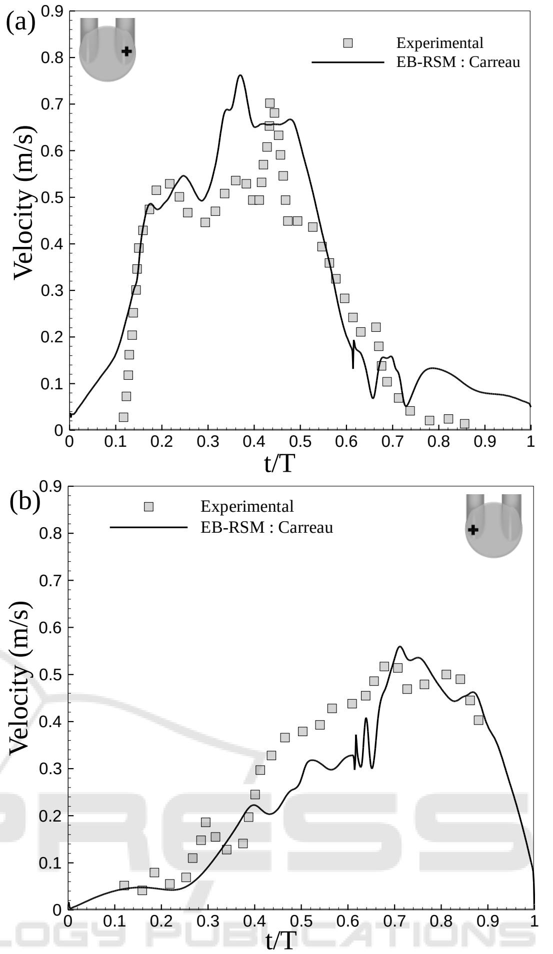

Figure 6: Variation of velocity magnitude at (a) mitral port and (b) aortic port.

using Pointwise CFD mesh generation software (V16.04R4) from Pointwise Inc. (Pointwise, 2011) and STAR-CCM+ 10.02 (StarCCM, 2015). Figure 2a shows the overset regions in an arbitrary position. As alluded above, the overset mesh methodology has been employed to simulate the valve closure. To achieve that a small gap between the background region and the valve should be left and the number of cells should be fewer than four. In all present work the number of cells is two and as a result the zero gap algorithm will apply; Figure 2b. These cells and the overset cells outside the background region will be inactive during the simulation, as shown in Figure 2c, which illustrates the mesh after starting the simulation. Figure 4 depicts the variation of y-velocity with mesh size for the EB-RSM model with the non-Newtonian Carreau model. A near wall resolution is assessed using the non-dimensional distance to the first near-wall grid point, $y^+ = \frac{y}{\mu(|S|)}\sqrt{\rho\tau_w}$; where y is the distance from the first cell centre to the wall, ρ is

Figure 7: Snapshots of stream lines originated from line 1 and 2.

the density of blood, $\mu(|S|)$ is the blood viscosity, and τ_w is the wall shear stress; where a prism mesh with six layers was used to resolve the boundary layer. In all simulations this value was set to the recommended value of $y^+ \approx 1$ for all locations inside the device. The mesh MO4 is selected for the following simulations, which is adequate to capture the properties of the flow within the chamber and near the valves.

For transient simulations, the sensitivity to time step was investigated. The Courant-Friedrichs-Lewy number $CFL = \frac{U\Delta t}{\Delta x}$ is employed during the simulation, where U is the local cell velocity and Δx is the characteristic cell length scale. Generally, in the zone of interest, CFL should be of the order of unity for unsteady analysis. In the present study, different time steps were tested with the same conditions, as a result, the time step $\Delta t = 0.001s$ was found to be satisfactory; resulting in a maximum CFL number of around 1 inside the chamber.

The numerical simulation was allowed to continue until a time periodic flow was obtained, in order to obtain a fully converged unsteady solution. In this study, the simulation was run for 5 complete pump flow cycles and the forth cycle has been chosen to extract the data from the simulation. Figure 5 illustrates the history of velocity magnitude at point in the chamber.

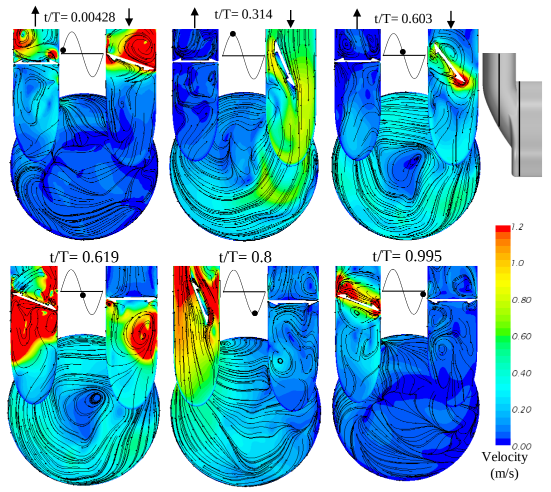

Figure 8: Contour of velocity magnitude at centre of the valves and $z/z_c = 0.21$ from the front face of the chamber.

5 RESULTS AND DISCUSSION

5.1 Velocity Distribution

The setup of the current numerical simulation was first validated by comparing the instantaneous flow fields in the device against the available PIV experimental data (Medvitz et al., 2009). The comparisons include traces of instantaneous velocity magnitude at extraction points in the chamber located at 25 percent of the chamber's radius from the wall on the plane $z/z_c = 0.159$ from the front face of the chamber.

Figure 6 depicts the comparison of velocity magnitude at two points within the chamber; proximal to the mitral port and to the aortic port as shown. The results from the EB-RSM turbulent model for the non-Newtonian blood viscosity model are presented in this figure. The RSM model has been chosen because it accounts for more realistic transport, or history, of three-dimensional effects as reported in our previous study (Al-Azawy et al., 2016).

From the figure, it can be seen that there ia good agreement with the experimental date, especially at the beginning of the diastolic phase at the point proximal to the mitral port, where the flow is injected into the chamber over the moving valve (see Figure 6a). This is with the notable exception of the Carreau model at location $t/T = 0.4$ where the velocity is higher. However, the simulation with valve rotation gives better prediction than assuming a constant open position. Again the velocity records a closer agreement with the experimental data at the aortic port, where the flow pumps towards the outlet port, during the systole (Figure 6b).

Figure 7 provides streamlines seeded from the inlet surface towards the chamber through the mitral valve. For the sake of clarity the streamlines originated from lines 1 and 2 in order to observe the complexity of the flow behaviour behind the valve (see line 1) and inside the chamber (see line 2).

Furthermore, to provide a more quantitative assessment of the blood flow within the device, the velocity contour through the valves and the chamber with streamlines are provided in Figure 8.

5.2 Clinical Issues

Figure 9 displays the flow behaviour and turbulent kinetic energy (TKE) at six points during the cycle, corresponding to early, peak and late in diastole and then systole. In this figure, the mesh configurations are displayed with the valve rotation, where the black mesh represents the background while the red mesh represents the overset mesh. It is expected that the flow behaviour and TKE would be highly sensitive to valve rotation. In addition, this investigation is significant as the turbulence effects cause potential damage to the blood cells. However, this is directly related to the thrombus and haemolysis which are the main causal risk associated with heart pumps.

6 CONCLUSIONS

Based on the validation with the experimental available data, it can be concluded that the overset mesh zero gap technique can be used successfully for the CFD simulation of blood pump. The present numerical study was conducted to describe the non-Newtonian, transient, and turbulent flow through an LVAD, using an elliptic blending Reynolds stress model. The overset mesh approach was employed for both pusher plate movement and valve rotation within the positive displacement pump. The zero gap technique was used to model the valve closure in order to mimic the natural scenario of heart pump. The TKE and velocity with streamlines were investigated in various positions of valve during the rotation.

ACKNOWLEDGEMENTS

The financial support from the Higher Committee for Education Development in Iraq and the University of Wasit is gratefully acknowledged. 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.

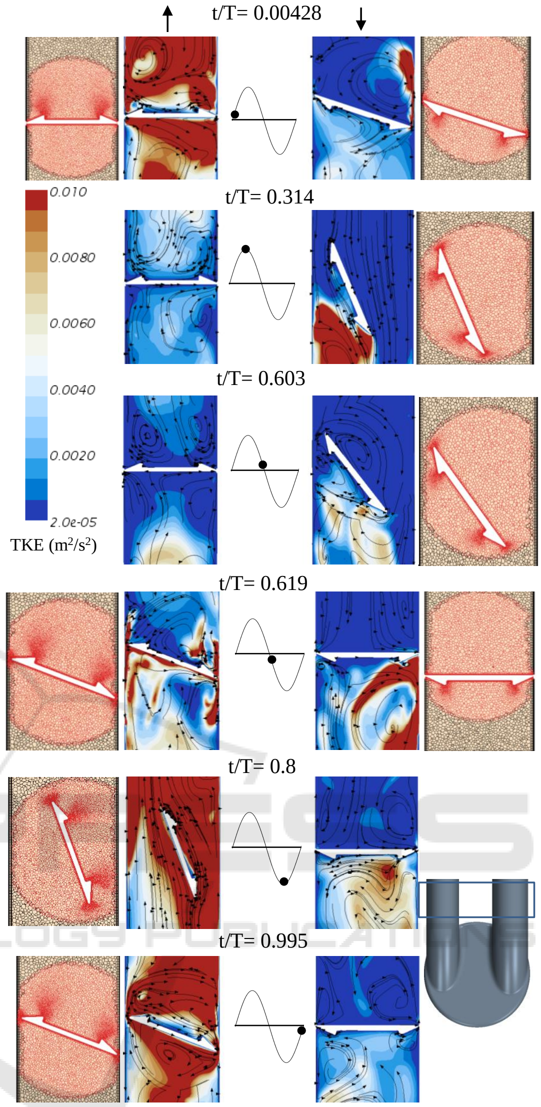

Figure 9: Snapshots of contour of turbulent kinetic energy with mesh configuration.

REFERENCES

Al-Azawy, M. G., Turan, A., and Revell, A. (2015). Investigating the Use of Turbulence Models for Flow Investigations in a Positive Displacement Ventricular Assist Device. *6th European Conference of the International Federation for Medical and Biological Engineering*, 45:395–398.

Al-Azawy, M. G., Turan, A., and Revell, A. (2016). Assessment of turbulence models for pulsatile flow inside a heart pump. *Computer Methods in Biomechanics and Biomedical Engineering*, 19(3):271–285. PMID: 25816074.

Avrahami, I. (2003). *The Effect of structure on the hemodynamics of artificial heart pumps*. PhD thesis, Tel-Aviv University.

Bluestein, D., Rambod, E., and Gharib, M. (2000). Vortex shedding as a mechanism for free emboli formation in mechanical heart valves. *Journal of biomechanical engineering*, 122(2):125–134.

Carreau, P. J. (1972). Rheological Equations from Molecular Network Theories. *Journal of Rheology*, 16(1):99–127.

Johnston, B. M., Johnston, P. R., Corney, S., and Kilpatrick, D. (2004). Non-Newtonian blood flow in human right coronary arteries: steady state simulations. *Journal of biomechanics*, 37(5):709–20.

Kiris, C., Kwak, D., Rogers, S., and Chang, I. (1997). Computational approach for probing the flow through artificial heart devices. *Journal of biomechanical engineering*, 119(4):452–460.

Kreider, J. W., Manning, K. B., Oley, L. a., Fontaine, A. a., and Deutsch, S. (2006). The 50cc Penn State left ventricular assist device: a parametric study of valve orientation flow dynamics. *ASAIO journal (American Society for Artificial Internal Organs : 1992)*, 52(2):123–31.

Medvitz, R. B. (2008). *Development and Validation of a Computational Fluid Dynamic Methodology for Pulsatile Blood Pump Design and Prediction of Thrombus Potential*. PhD thesis, Pennsylvania State University, University Park, PA.

Medvitz, R. B., Kreider, J. W., Manning, K. B., Fontaine, A. A., Deutsch, S., and Paterson, E. G. (2007). Development and validation of a computational fluid dynamics methodology for simulation of pulsatile left ventricular assist devices. *ASAIO journal (American Society for Artificial Internal Organs : 1992)*, 53(2):122–131.

Medvitz, R. B., Reddy, V., Deutsch, S., Manning, K. B., and Paterson, E. G. (2009). Validation of a CFD methodology for positive displacement LVAD analysis using PIV data. *Journal of biomechanical engineering*, 131(11):111009 1–9.

Peskin, C. S. (2002). The immersed boundary method. *Acta Numerica*, 11:479–517.

Pointwise (2011). Pointwise, Inc. Release 16.04R4 .

StarCCM (2015). CD-Adapco User Guide, STAR-CCM+ Version 10.02 .

Stijnen, J., de Hart, J., Bovendeerd, P., and van de Vosse, F. (2004). Evaluation of a fictitious domain method for predicting dynamic response of mechanical heart valves. *Journal of Fluids and Structures*, 19(6):835–850.

Stijnen, J. M. A. (2004). *Interaction between the mitral and aortic heart valve an experimental and computational study*. PhD thesis, Eindhoven University.

Yin, W., Alemu, Y., Affeld, K., Jesty, J., and Bluestein, D. (2004). Flow-induced platelet activation in bileaflet and monoleaflet mechanical heart valves. *Annals of biomedical engineering*, 32(8):1058–1066.