Fifth International Conference on CFD in the Process Industries
CSIRO, Melbourne, Australia
13-15 December 2006

HEMODYNAMIC STUDY IN CAVOPULMONARY VASCULAR SYSTEM BY CHARACTERISTIC BASED SPLIT WITH ARTIFICIAL COMPRESSIBILITY SCHEME

K. CHITRA[1], S.VENGADESAN[2], T. SUNDARARAJAN[3] and P. NITHIARASU[4]

[1]Research Scholar, Dept. of Applied Mechanics, IIT Madras, Chennai, India.
[2]Assistant Professor, Dept. of Applied Mechanics, IIT Madras, Chennai, India.
[3]Professor, Dept. of Mechanical Engineering, IIT Madras, Chennai, India.
[4]Reader, Dept. of Civil and Computational Engineering,
University of Wales Swansea, Swansea UK.

ABSTRACT

An Artificial Compressibility Characteristic Based Split Scheme (CBS-AC) employed in FEM is used to predict the flow pattern in Cavopulmonary vascular system. Such type of flow is very complex due to varying nature of flow through SVC (Super Vena Cava) and IVC (Inferior Vena Cava), confluence at T-junction, recirculation etc,. In this work, numerical simulations are performed to extend the applicability of the present scheme and to understand initially the flow pattern in the TCPC (Total Cavopulmonary connection) following Fontan surgical procedure. It is further intended to estimate energy losses and evaluate hemodynamic stresses. The basic code is validated against benchmark data and then applied to study the present problem. The results are compared with published results on a similar configuration.

Keywords: FEM, CBS-AC, Hemodynamic stresses, Fontan procedure, TCPC.

NOMENCLATURE

x characteristic length
P pressure
$\boldsymbol{U}$ velocity
ρ density
μ dynamic viscosity
t time
τ shear stress
c wave speed

INTRODUCTION

The human vascular system comprises the heart and blood vessels, and functions in such a way as to supply each organ with a certain amount of blood, which may vary depending on physiological conditions and organ demands. Hemodynamic factors are hypothesized to be important in normal adaptive response of blood vessels to chronic changes in physiological demands and in maladaptive responses leading to vascular disease (Fung Y.C., 1984). Specifically, it is hypothesized that flow recirculation, high particle residence time, and low wall shear stress are responsible for the localization of atherosclerotic plaques in regions of complex flow in the arteries. Considering the strong correlation between the localization of atherosclerosis and arterial wall shear stress, quantifying wall shear stress is important to understand the development of arterial disease. In most prior investigations, computational and in-vitro models have been used to quantify shear stress. The inadvertent introduction of such regions of elevated blood residence time in the design of interventional devices and prostheses (e.g. artificial hearts and valves, stents, bypass grafts), might lead to their ultimate failure. Computational fluid dynamic (CFD) modelling techniques are useful in studying the shear stresses in re-circulating and stagnant zones of such flows (Chiu *et al*. 1998; Perktold and Rappitsch, 1994; Taylor, 1996). Hence, once developed sufficiently, CFD models are useful in studying these complex bio-fluid dynamic problems.

An artificial compressibility scheme (AC) using finite element method is proposed in the present study. The Characteristics Based Split (CBS) scheme is fully explicit and accurate as well as effective in solving incompressible fluid dynamics problems. As this scheme includes split and velocity correction, it utilizes good features of both velocity correction and standard artificial compressibility schemes. Explicit CBS scheme, however, avoids the solution of matrices arising from the discretization of Poisson type equations by introducing the artificial compressibility (AC) concept together with velocity correction (Nithiarasu, 2003). It is convenient to use unstructured mesh due to explicit scheme. Hence, the CBS-AC is adopted for the Bio-Fluid simulation in a Cavopulmonary vascular system. In the present work, an attempt has been made to understand the flow pattern and correlate the hemodynamics for various flow parameters.

NUMERICAL SOLUTION OF GOVERNING EQUATIONS

Governing Equation

The non-dimensional forms of the equations are:

Continuity equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial U_i}{\partial x_i} = 0 \qquad (1)$$

and the vector momentum equation

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j}(u_j U_i) = -\frac{\partial P}{\partial x_i} + \frac{1}{\mathrm{Re}}\frac{\partial \tau_{ij}}{\partial x_j} \qquad (2)$$

Where,

$$\tau_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij} \tag{3}$$

and *Re* is the Reynolds number.

In the above equations, δ_{ij} is the Kroneker delta and is equal to unity when $i=j$; and zero when $i \neq j$, u_i are the velocity components; ρ density; p is the pressure.

An artificial compressibility scheme employing finite element method, known as the Characteristics Based Split (CBS) scheme, has been adopted (Nithiarasu P., (2003, 2004, 2005), Nithiarasu *et al*., 2004, Zienkiewicz *et al*., 2005). This scheme includes split and velocity correction, and an artificial compressibility term. The scheme is fully explicit. Explicit time stepping schemes are popular and have advantages in both compressible and incompressible fluid dynamic studies due to their simplicity in solution. The AC-CBS scheme is used for the simulation of Bio-fluid flows and to study the hemostatisis of dynamic quantities. The scheme essentially contains three steps. In the first step, an intermediate velocity field is established: in the second step, the pressure is obtained from a pressure (continuity) equation and finally the intermediate velocities are corrected to get the final velocities.

Step 1: Intermediate momentum

$$\Delta\tilde{U} = \tilde{U}_i - U_i^n = \Delta t\left(\frac{\partial}{\partial x_j}(u_j U_i) + \frac{1}{\mathrm{Re}}\frac{\partial \tau_{ij}}{\partial x_j} + \frac{\Delta t}{2}u_k\frac{\partial}{\partial x_k}\left(\frac{\partial}{\partial x_j}(u_j U_i)\right)\right)^n \tag{4}$$

Step 2: Density or Pressure

$$\Delta\rho = \rho^{n+1} - \rho^n = \left(\frac{1}{c^2}\right)^n \Delta p = \left(\frac{1}{c^2}\right)^n (p^{n+1} - p^n) = -\Delta t\left[\frac{\partial U_i^n}{\partial x_i} + \theta_1\frac{\partial \Delta\tilde{U}_i}{\partial x_i} - \Delta t\theta_1\left(\frac{\partial^2 p^n}{\partial x_i \partial x_i} + \theta_2\frac{\partial^2 \Delta p}{\partial x_i \partial x_i}\right)\right] \tag{5}$$

Step 3: Momentum correction

$$\Delta U_i = U_i^{n+1} - U_i^n = \Delta\tilde{U}_i - \Delta t\,\frac{\partial p}{\partial x_i}^{n+\theta_2} \tag{6}$$

Where, $0.5 \leq \theta_1 \leq 1$ and $0 \leq \theta_2 \leq 1$. The above equations are derived from the observation made that the time discretization is carried out along the characteristic. In equation (5), the wave speed c is replaced with an appropriate artificial compressibility parameter β according to Nithiarasu (2003). A simple approximate integration backwards gives the above equations with added extra convection stabilization terms (last term in RHS at step1). These extra terms are consistent and suppress any oscillations due to highly convective flows.

An unstructured mesh with triangular elements is used to discretize the flow domain. The time step is chosen to satisfy the Courant condition for integration. The scheme is accurate to second order. The model has been developed based on the above formulation and solution methodology, and tested for the steady and transient flow problems. The results are comparable with available published solutions.

Validation of the code

In order to validate the numerical scheme, the benchmark problem of laminar 2-D flow over a backward facing step is considered. Numerical simulations were carried out for a Reynolds number of 229. Along the wall no-slip condition is used with parabolic velocity profile at the inlet. A Drichlet type boundary condition for pressure was used for the outlet. The result is compared with the experimental result (Denham *et al*. 1973) and is shown in Fig. 1. The comparison shown in Fig.1 depicts good correlation with experimental data. As the flow develops, the inconsistencies in comparisons are damped out and the fully developed flow compares very well with the simulations. Considering the advantages of CBS scheme over the other methods (Nithiarasu P, 2003), the results show a considerable advantage over many currently available schemes.

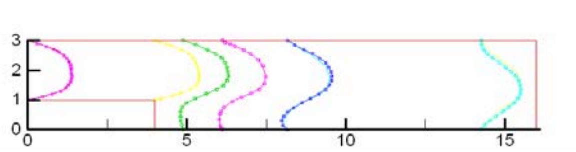

Figure 1. Comparison of computed velocity distribution (solid line) at different sections with experimental data [Denham *et al.* 1973](circles).

Hemodynamic studies

Flow in an expansion chamber is studied as a first stage for the computation of hemodynamic quantities in a representative hemodynamic system. Then the model has been extended to study the flow pattern in the Cavopulmonary vascular system.

Expansion Flow Chamber

Truskey *et al.,* (1995) used an asymmetric sudden expansion flow chamber in order to examine the response of endothelial cells to spatially non-uniform flows. The experimental results of shear stress and recirculation zone were compared with two and three-dimensional numerical solutions. The same problem is considered here for the computation of shear stress in the recirculation zone and compared with the existing solutions.

An unstructured mesh with triangular elements is used to discretize the study region. Grid size at the boundary and interior with growth rate of 1.1 is used to capture the boundary layer effect. The inlet boundary is considered fully developed with known velocity profile. The flow is 5cm^3/s with an entrance length of 3.0cm. Along the lateral side no-slip boundary condition is used. Solutions of the CBS-AC scheme are compared with the experimental data and numerical results (Truskey *et al*., 1995) for an expansion ratio of 2.37. The non-dimensionalized geometry for the expansion flow chamber and flow fields for the different Reynolds numbers are shown in Fig. 2a. The corresponding non-dimensionalized length scales of recirculation zone are comparable with existing solution are shown in Fig.2b. Figure 2c shows the variation of shear stress along the re-circulation zone for *Re*=473. Fig. 2c shows a marginal departure from existing solution. This could have been due to use of fine mesh in the present study in comparison with Truskey *et al*., (1995).

In Fig. 2c shear stress in the recirculation is plotted in dimensional values for comparison with an existing solution. With these validations the model has been extended to simulate the flow pattern in the cavopulmonary vascular system.

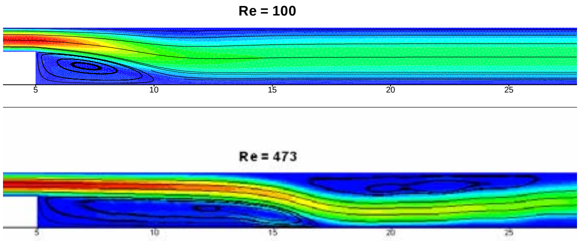

Figure 2a. Stream traces for the different Reynolds number (Number of elements =14847, Nodes =7876)

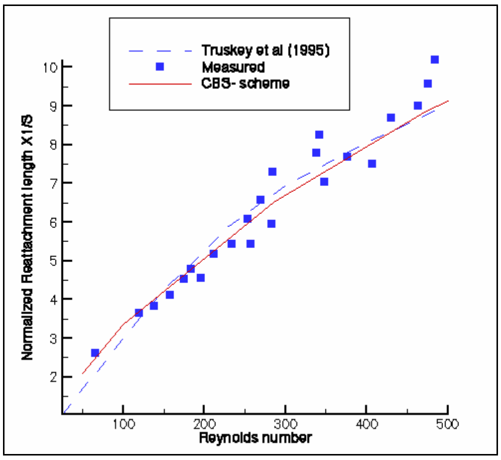

Figure 2b. Comparison of the size of recirculation zone for an expansion ratio of 2.37.

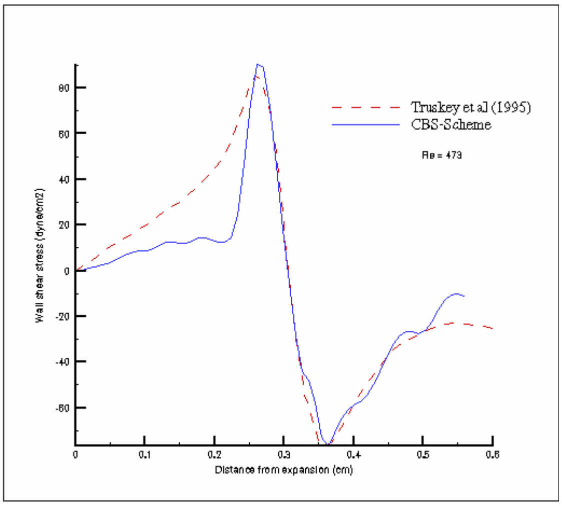

Figure 2c. Distribution of wall shear stress along the recirculation zone for Re=473 with an expansion ratio of 2.37.

Total Cavopulmonary Connection (TCPC)

The CBS code, as validated above, has been used to carry out numerical simulation of blood flow to the lung after a surgical procedure (TCPC - Fontan procedure). Fontan procedure concerns blood flow distribution in the TCPC as shown in Fig.3. According to the anatomy of the TCPC, blood flows from the SVC (superior vena cava) and IVC (inferior vena cava) and confluent at the T-junction. This is followed by a downstream flow into the left pulmonary artery (LPA) and right pulmonary artery (RPA). Downstream from the confluence of the SVC and IVC, the main pulmonary artery bifurcates into four branches, namely, the left upper pulmonary artery (LUPA), left lower pulmonary artery (LLPA), right upper pulmonary artery (RUPA), and right lower pulmonary artery (RLPA). These branch flows proceed further downstream to the left and right lungs. For better flow distribution and energy preservation, surgical reasoning suggests the enlargement of IVC and SVC anastomoses. As the main pulmonary is approached, the diameter of the SVC and IVC increase gradually, reaching 130% diameter enlargement at the IVC and SVC anastomoses. As clinical reports indicate, the SVC, carrying approximately one third of the systemic venous return, goes preferentially to the larger right lung. As a result, the IVC flow carries two-thirds of the systemic venous return which goes to the smaller left lung. In this study the focus is to see the effect of the offset on two important energy indexes, the hydraulic power, and the total energy loss coefficient. The axes of symmetry of both the IVC and SVC are kept normal to the pulmonary artery. This creates the stagnation point at the junction, and avoids the smooth flow from SVC-IVC to lungs. Hence, to achieve the smooth flow pattern different offset sizes are used between SVC-IVC for flow simulation. The offset size should be decided based on minimum energy loss. Numerical study is carried out to optimize the offset size in TCPC surgical procedure, based on reduced energy loss. A similar study was carried with a different methodology by Sheu *et al*. (1999).

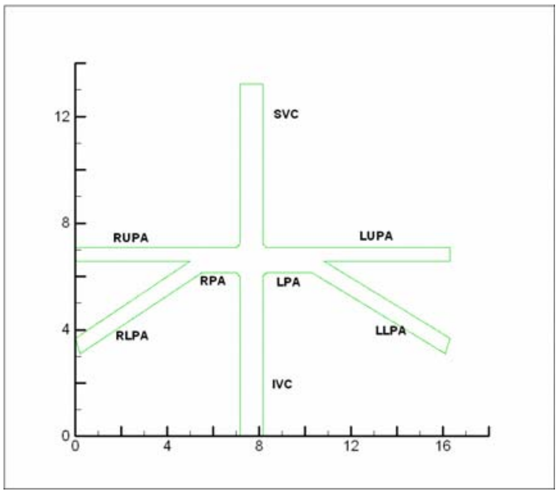

Figure 3. TCPC connection for 0 offset
Number of elements =141170, Nodes =72277
SVC- Superior Vena Cava
IVC- Inferior Vena Cava
RPA- Right Pulmonary Artery
LPA- Left Pulmonary Artery

The blood flow is assumed to be Newtonian. Two-dimensional incompressible flow is considered. The vessel walls are considered rigid for simplicity. The non-dimensionalised form of the Navier-Stokes equations are solved using the CBS scheme. The parabolic velocity profiles are used as input at SVC and IVC. The maximum inlet velocities of the IVC and SVC are 0.256m/s and

0.128m/s respectively. This corresponds to the values used by Sheu *et al.* (1999). Along the wall boundary, no-slip condition is used. Flow patterns in terms of streamlines from the present simulations for different offsets are shown in figure 3a & 3b. The velocity field and visualized flow structures provide an understanding of local flow behaviour including the size and locations of flow separation and reattachments, flow stagnation regions, secondary flows, and recirculatory flow regions.

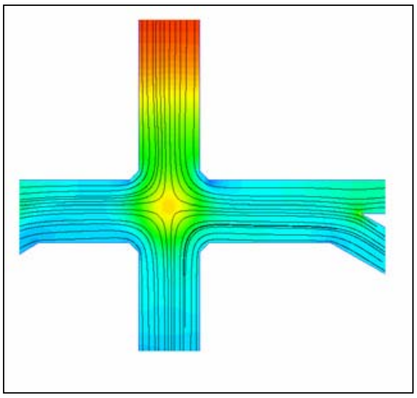

Figure 3a shows the stream traces for 0 offset of SVC-IVC. A clear stagnation point is formed within the confluence region where SVC and IVC flows meet. As stagnation region is one flow structure that appeared to affect pressure drop significantly. The stagnation region is an area of high pressure, usually found where flow impingement occurs; large areas of stagnation can create increased pressure losses since cross-sectional forward flow area at the stagnation region may decrease, which in turn causes local velocities to increase and local pressure to decrease. The stagnation region causes, a diversion of the IVC and SVC flow into the pulmonary arteries. As the SVC-IVC flow rate ratio decreased, the stagnation area moved toward the SVC as a consequence of the increased flow energy within the IVC flow stream. More work is therefore needed to maintain the flow through the SVC and connection area; this resulted in increased overall pressure drop. To avoid this situation and to have a smooth flow with minimized energy loss SVC is connected to IVC with different offsets. Flow pattern for different parameters are shown in Fig. 3b. The optimum offset size should be decided based on minimum energy loss.

Figure 3a. Predicted Stream traces for 0 offset of SVC-IVC connection

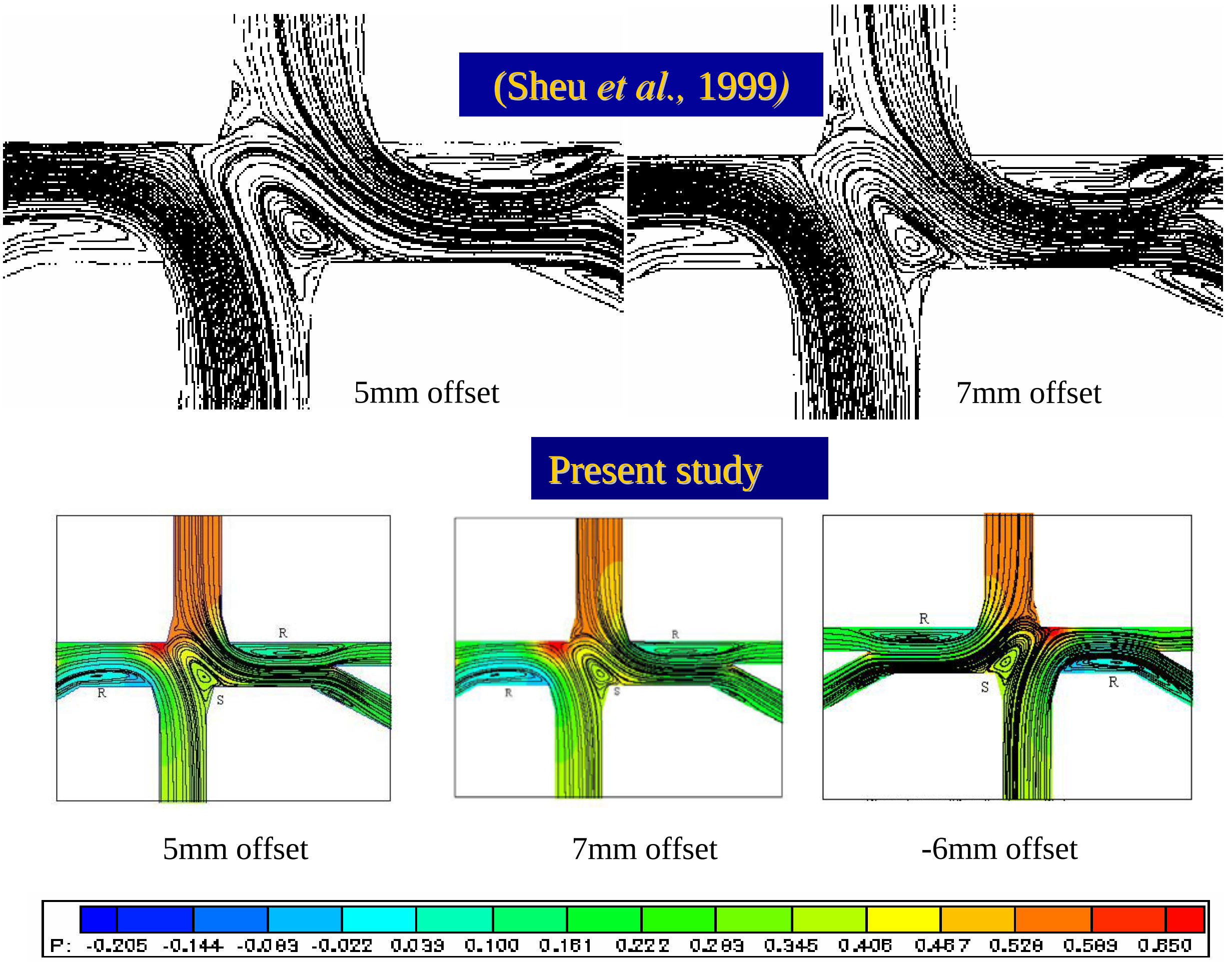

Figure. 3b Flow pattern with stream traces in Cavopulmonary connection for the offset 5mm, 7mm and -6mm offsets. S-Stagnation point, R-Re-circulation.

Table 1 shows the energy loss coefficient for different parameters. Based on these computations, zero offset gives more energy loss than compared to other offsets as predicted by Sheu *et al.,* (1999). Hence, the offsets of 7mm would be more suitable for SVC-IVC connection. Numerical simulation study helps to understand the flow pattern for various parameters. The flow pattern indicates the stagnation point moved down at the corner of LPA (Fig.3b) for SVC-IVC with offsets.

Offsets for SVC-IVC connection	Coefficient of Energy in %
0 mm	73.176
5 mm	75.501
7 mm	76.611
-6 mm	75.137

Table-1. Coefficient of energy for different offsets

In summary, the presence of recirculation regions within the confluence has implications for pressure drop. Strong recirculating flows would cause energy losses and thereby lead to increased pressure drop. Figure 3b shows the flow pattern for 5mm, 7mm and -6mm offsets of SVC with IVC. The recirculation regions and the flow pattern predicted by the present work have been compared with earlier study (Fig. 3b, Sheu *et al*. 1999). Herein, the focus is to investigate the effect of SVC-IVC offsets in terms of the energy loss and other hydrodynamic characteristics of the flow pattern. Figure 3c show the non-dimensional wall shear stress along right and left pulmonary artery for 7mm offset. The wall shear stress for 7mm offset is shown because of minimum energy loss coefficient is obtained (Table-1) compared to other parameters. Further interest is to study the hemodynamics. It is observed that, Fig. 3c, indicates the clear variation of low shear stress close to re-circulation stagnant flow and high oscillatory shear close to re-attachment point. This emphasises the direct hemodynamic correlative studies for localization of vascular disease for assumptions close to real situations. The rheologic properties of blood make it a non-Newtonian fluid, with viscosity depending on vessel diameter and shear rate. However, in large vessels blood behaviour can be considered as Newtonian (Cheng *et al*., 2002). As a first approximation, Newtonian, rigid wall with fully developed inflow conditions are used. In future, the same problem will be studied for a non-Newtonian with pulsatile motion to understand the hemodynamics.

These recirculation regions and reattachment points are of interest as they are responsible for thrombus formation and the present code predicts these regions reasonably well.

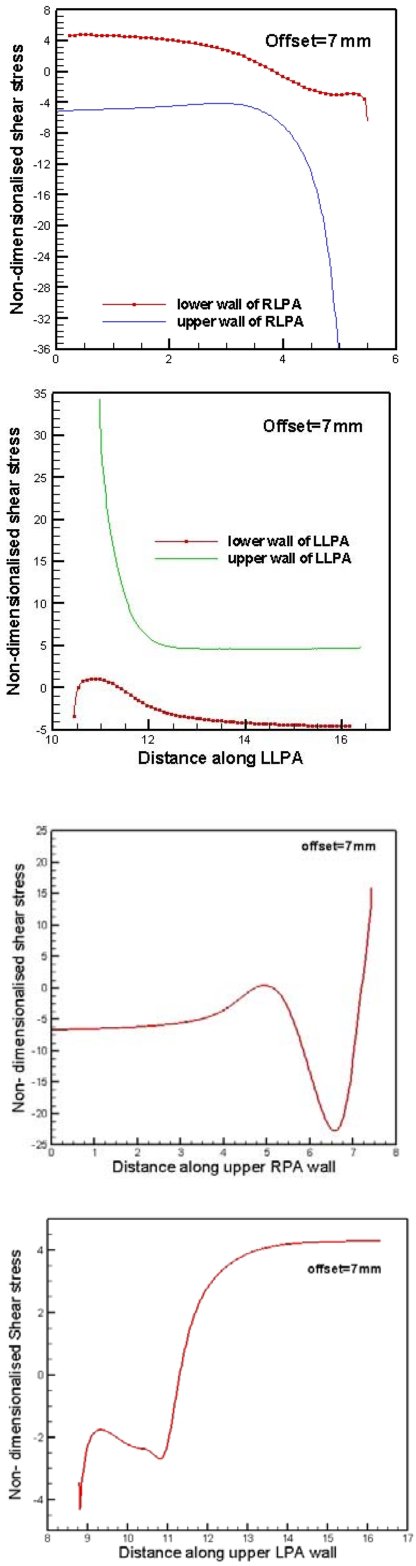

Figure 3c. Non-dimensional shear stress along the wall.

CONCLUDING SUMMARY

The CBS-AC scheme algorithm has been tested for steady and unsteady flow problems (Nithiarasu P., 2003). The same formulation has been used in the present study for the simulation of Bio-Fluid flows. The model extended further for estimating quantitative hemodynamic characteristics such as shear stress and velocity field in recirculation zone. The different offsets of SVC-IVC connection are used for simulation. The results indicate good prediction of flow pattern in the Cavopulmonary arteries using CBS algorithm. The same model can now be used for the prediction of hemodynamic quantities which correlates with vascular disease in the Cavopulmonary vascular system.

REFERENCES

CHENG C.P, David PARKER, and TAYLOR C.A., 2002, "Quantification of large blood vessels using Lagrangian interpolation functions with Cine Phase-Contrast Magnetic Resonance Imaging", Annals of Biomedical Engineering, 30, pp. 1020-1032.

CHIU J-J, WANG DL, CHIEN S, SKALAK R, and USAMI S., 1998, "Effects of disturbed flow on endothelial cells", ASME J. Biomech. Eng., 120, pp. 2-8.

DENHAM M. K. and PATRIK M.A., 1974, "Laminar flow over a downstream-facing in a two-dimensional flow channel", Transactions of the Institution of Chemical Engineers, 52, pp. 361-367.

FUNG YC, 1984, "Biodynamics": Circulation. New York: Springer-Verlag.

NITHIARASU, P., "An efficient Artificial Compressibility (AC) scheme based on the Characteristic Based Split (CBS) method for incompressible flows", Int. Journal for Numerical Methods in Engineering, 56, 1815-1845, 2003.

NITHIARASU, P., MATHUR, J.S., WEATHERILL, N.P. and Morgan, K., "Three dimensional incompressible flow calculations using the Characteristic Based Split (CBS) scheme", Int. Journal for Numerical Methods in Fluids, 44, 1207-1229, 2004.

NITHIARASU, P., "A fully explicit Characteristic Based Split (CBS) scheme for viscoelastic flow calculations", Int. Journal for Numerical Methods in Engineering, 60, 949-978, 2004.

NITHIARASU, P. and LIU C-B, "Steady and unstable flow calculations in a double driven cavity using the explicit CBS scheme", International Journal for Numerical Methods in Engineering, 63, 380-397, 2005.

PERKTOLD and RAPPITSCH K. PERKTOLD and RAPPITSCH G., 1995, "Computer simulation of local blood flow and vessel mechanics in a compliant carotid artery bifurcation model", J. of Biomechanics 28, pp. 845-856.

SHEU T.W.H., TSAI S.F., HWANG W.S. and CHANG T.M, 1999, "A finite element study of the blood flow in total cavopulmonary connection", Computers & Fluids, 28, pp. 19-39.

TRUSKEY, G. A., BARBER, K. M., ROBEY, T. C., OLIVIER, L. A., and COMBS, M. P., 1995, "Characterization of a Sudden Expansion Flow Chamber to Study the Response of Endothelium to Flow Recirculation," ASME J. Biomech. Eng., 117, pp. 203-210.

TAYLOR CA, HUGHES TJ, ZARINS CK., 1996, "Computational investigations of vascular disease", Computers in Physics 10, pp. 224-232.

ZIENKIEWICZ O.C., TAYLOR R.L. and NITHIARASU P., 2005, "The finite element method for fluid Dynamics", Elsevier.