NUMERICAL MODELING OF CARDIOVASCULAR FLOWS: INTEGRATING HIGH RESOLUTION CFD & EXPERIMENTAL TECHNIQUES[1]

Liang Ge,[1] Kerem Pekkan,[2] Hwa L. Leo,[2] Diane de Zelicourt,[2] Fotis Sotiropoulos[1] & Ajit Yoganathan[2]
[1]School of Civil and Environmental Engineering, Georgia Tech, Atlanta, GA, USA
[2]Wallace Coulter School of Biomedical Engineering, Georgia Tech, Atlanta, GA, USA

Computational Fluid Dynamics modeling of cardiovascular flows first began some 30 years ago with the pioneering work by Peskin and co-workers. In recent years the advent of powerful and affordable computational platforms along with the development and proliferation of advanced commercial and academic software have contributed to the emergence of CFD as a useful bioengineering research tool alongside with in vitro and in vivo studies. Cardiovascular flows take place in complex, multi-connected domains with compliant walls and flexible immersed boundaries and are dominated, among others, by pulsatile effects, 3D separation and vortex formation, regions of flow reversal, periodic transition to turbulence and laminarization, and non-Newtonian effects. These complexities pose unique modeling challenges and necessitate a close synergy and integration between CFD modelers and experimentalists to guide model development and validation. In this paper we report recent progress toward the development and validation of a high resolution numerical method capable of quantitatively accurate predictions of complex cardiovascular flows. The method employs domain decomposition with body-fitted, domain-structured, Chimera overset grids to discretize arbitrarily complex, multi-connected geometries. The 3D, unsteady Navier-Stokes equations are solved with a second-order accurate, dual-time stepping artificial compressibility approach [1]. We present results for two flow cases, each representative of different kinds of modeling challenges: 1) flow through a bileaflet mechanical heart valve with the leaflets fixed at the fully open position; and 2) flow through an anatomically realistic Total Cavopulmonary Connection (TCPC). For both cases the numerical simulations and laboratory experiments are carried out concurrently. Preliminary findings from the former are used to define experimental data needs and guide the experiments by identifying areas in the flow where complex phenomena occur. High resolution measurements are then carried out to collect comprehensive data sets for validating the numerical model.

Simulations of Bileaflet Mechanical Heart Valve Flows

Approximately 170,000 individuals worldwide receive prosthetic heart valves every year and over half receive mechanical heart valves (MHVs). Recipients of MHVs, however, must take anticoagulant medication because of the potential for thromboembolic complications. Such complications are believed to be caused by high blood shear stresses, turbulence, and the overall complexity of hemodynamics in MHV. CFD modeling of MHVs promises to provide designers with a tool to refine existing MHV designs; however, the information provided by CFD must be accurate and reliable. We carry out calculations for a typical MHV geometry (Fig. 1a) without adopting simplifying symmetry assumptions on grids that are at least one order of magnitude finer than those used in previously reported studies in the literature—typically one to two million grid nodes are used. These computations illustrate clearly the highly 3D structure of the flow (see Fig. 1b), question the validity of computationally expedient assumptions of flow symmetry, and demonstrate the need for highly resolved, fully 3D simulations if CFD is to accurately predict the flow in MHV [1]. The numerical findings stimulated a series of laboratory experiments specifically tailored to confirm the computational results and to provide data for CFD validation. The experiments were conducted for a valve/aorta configuration identical to that used in the CFD model using Particle Image Velocimetry (PIV). As shown in Fig. 1c the computed time averaged velocity profiles are in very good agreement with the measurements. Comparisons over a range of Reynolds numbers will be presented at the conference to establish the accuracy of our numerical method and help elucidate the rich dynamics of the flow downstream of the leaflets with increasing Reynolds number.

Flow in a Total Cavopulmonary Connection Configuration

The single ventricle is a congenital heart defect in which the right side of the heart is hypoplastic or totally absent. This anomaly results in mixing of the oxygenated and deoxygenated blood in the single ventricle, reducing the amount of oxygen transferred to the body. In U.S. two in 1000 babies are born with a single ventricle heart defect. Palliative surgical treatments are performed in stages as the child grows. Introduced by de Leval et al in 1988 [2], the final stage is the TCPC, which bypasses the right side of the heart and the single ventricle drives blood throughout the pulmonary and systemic circulations (see Fig. 2a). Previous investigations have shown that the energetic efficiency of the connection is critical for long-term success following TCPC operations [3]. In this work we seek to develop a practical analysis and design tool for pre-surgical optimization of TCPC anatomies using both in vitro experimental and CFD modeling. Laboratory flow visualization experiments for a typical Fontan anatomy have underscored the enormous complexity of the flow even for flow rates in the laminar flow regime (Re ~ 300 to 1800). These experiments show that in the stagnation region where the IVC and SVC streams collide the flow is dominated by complex, unsteady, and highly 3D instabilities. Numerical simulations with the commercial code FIDAP have failed to reproduce these complexities, yielding steady and

---

[1] Sponsored by NIH grants RO1-HL-07262 and RO1 HL 67622

smooth flow in the stagnation region. Highly resolved, unsteady simulations using the numerical method of [1] have been able to yield flowfields, which exhibit very complex unsteady patterns similar to those observed in the experiments (see Fig. 2). Results from these simulations and comparisons with experiments will be presented at the conference

Closing Remarks and Future Research Directions

The examples presented above point to the need for an integrated, two-pronged modeling strategy, which combines both high resolution experimental techniques with high resolutions numerics. The validation results for the MHV case are promising as they suggest that our method can be used as the basis for developing a predictive CFD methodology for cardiovascular flows. Before this goal can be realized, however, much remains to be done. Our ongoing research focuses on the following topics: 1) simulation of pulsatile flows in domains bounded with complaint walls and containing flexible immersed boundaries; 2) grid embedding strategies for modeling small scale geometrical features, such as the hinge region in MHV; 3) development and validation of turbulence models capable of simulating pulsatile flows periodically undergoing transition to turbulence and laminarization; and 4) implementation of non-Newtonian effects into the model.

References

[1] Ge, L., Jones, S. C., Sotiropoulos, F., Healy, T., and Yoganathan, A. “Numerical Simulation of Flow in Mechanical Heart Valves: Grid Resolution and Flow Symmetry ,” in press, *ASME Journal of Biomechanical Engineering*, 2003.

[2] de Leval, M. R., Kilner, P., Gewillig, M., and Bull, C., 1988, Total Cavopulmonary Connection: A Logical Alternative to Atriopulmonary Connection for Complex Fontan Operations. Experimental Studies and Early Clinical Experience, J. Thorac. Cardiovasc. Surg., 96, pp. 682-695.

[3] Ryu, K., Healy, T. M., Ensley, A. E., Sharma, S., Lucas, C., and Yoganathan, A. P., 2001, Importance of Accurate Geometry in the Study of the Total Cavopulmonary Connection: Computational Simulations and In vitro Experiments, Ann. Biomedi. Eng., Vol. 29, pp. 844-853.

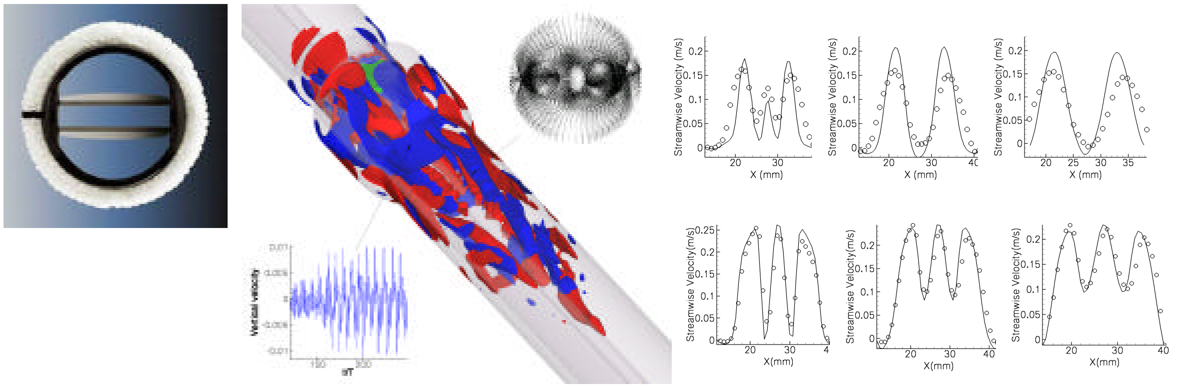

Figure 1. Direct numerical simulation of flow in a bileaflet MHV with the leaflets fixed in the fully open position (Re = 750). Overset grids with 1.5 million nodes have been used to discretize the leaflets and the model aorta. Left: Typical bileaflet valve; Center: Calculated instantaneous iso-surfaces of streamwise vorticity, cross-flow vectors, and typical time history; Right Measured and computed time-average streamwise velocity profiles at two horizontal planes (top and bottom) and at three streamwise locations downstream of the leaflets (left to right).

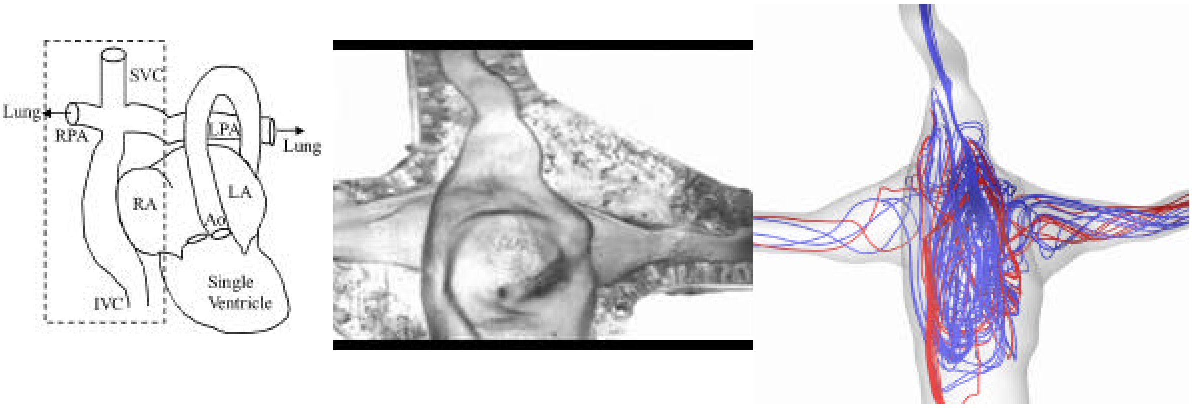

Figure 2. Total Cavopulmonary Connection (TCPC). Left: Schematic showing reconstructed circulation resulting from TCPC surgery. The dashed box highlights the reconstructed region. Center: Dye visualization of IVC flow in an anatomically realistic TCPC reconstructed from MRI data. Right: Simulated instantaneous streamlines showing the complexity and intense mixing of the SVC (blue traces) and IVC (red traces) flows.