A TIGHTLY COUPLED MULTI-SCALE CFD ANALYSIS OF THE FONTAN CIRCULATION

Marcus Ni[1], Alain Kassab[1], Eduardo Divo[1,2], William M. DeCampli[3,4] and Ricardo Argueta-Morales[3]

[1]Mechanical and Aerospace Engineering, University of Central Florida, Orlando, FL, USA
[2]Mechanical Engineering, Embry-Riddle Aeronautical University, Daytona Beach, FL, USA
[3]The Heart Center, Arnold Palmer Hospital for Children, Orlando, FL, USA
[4]College of Medicine, University of Central Florida, Orlando, FL, USA

The Fontan circulation is a result of the last (third stage) surgical procedure to correct a single ventricle congenital cardiac disorder in children. Although the Fontan circulation has been successfully established in surgeries over the years, it is flawed and can lead in certain cases to pre-mature death. The main cause of this failure is due to increased pulmonary vascular resistance. In healthy circulations the heart pumps directly to the lungs, where as “Single Ventricle” patients must use a single sided heart to supply blood to the rest of the body before the lungs. Improvements to the Fontan circulation have been proposed, but they require extensive care or external devices. In the analysis of the Fontan, ascertaining energy losses due to flow jet impingements and flow mixing is critical. Moreover, in order to better understand surgical alternatives is it important to have a robust multi-scale 0D-3D CFD analysis tool that permits investigation of surgical alternatives in a virtual physics-based environment. To this end, an-open loop lumped parameter model (LPM) is tightly coupled at the time step level with a full 3D Computational Fluid Dynamics (CFD) model, Figure 1. Using this model scheme, the Fontan conduit, Left-Figure 1, is no longer being modeled by the LPM and therefore not limited by the 0D nature of the vascular resistance, capacitance and inertia bed model. The CFD is able to take over at the area of interest which account for flow directionality and momentum transfer that the LPM is unable to capture. Through these studies we will predict circulation changes that will improve the Fontan circulation. We will present details of our modeling strategies as well as results from our flow simulations