

# Designing with Heart

CFD-based design optimization for a miniature ventricular assist implant can shave years off the medical device development cycle.

By Jingchun Wu, LaunchPoint Technologies, Inc., California, U.S.A. and Harvey Borovetz, McGowan Institute for Regenerative Medicine Pennsylvania, U.S.A.

An important challenge facing the design of turbodynamic ventricular assist devices (VADs) intended for long-term cardiac support is the optimization of the flow geometry to maximize hydraulic efficiency while minimizing the peak shear stress in the blood flow. High efficiency reduces the required battery size while low shear reduces the number of red blood cells that are ruptured by the pump. A pediatric heart-assist pump is particularly challenging. Due to its small size (about 28 mm diameter by 51 mm length), the design laws for adult-sized pumps do not apply, and they cannot be scaled. Therefore, the design of pediatric blood pumps must rely on modern design approaches to optimize the flow path. Computational fluid dynamics (CFD) has been widely used in the field of artificial heart pumps for the analysis of internal flow because it offers an inexpensive and rapid means of acquiring detailed flow field information that is expensive and painstaking through in vitro testing. LaunchPoint Technologies, Inc., in the United States, which developed the first magnetically levitated (maglev) heart pump (the Streamliner ventricular assist device that reached animal trials in 1998), finds that CFD is a powerful tool in the performance assessment and optimization of artificial heart pumps.

LaunchPoint has developed a CFD-based design optimization approach

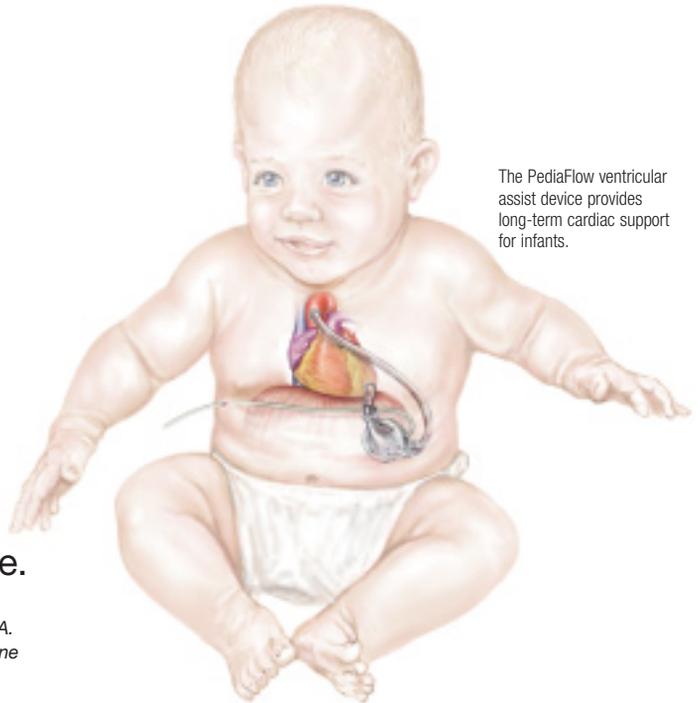
that integrates internally developed 3-D inverse blade design methods, parameterized geometry models, automatic mesh generators and mathematical models of blood damage with the commercial ANSYS CFX solver. The system provides rapid optimization for various types of centrifugal, mixed-flow and axial-flow blood pumps. The ANSYS CFX solver was chosen because of its robustness for computations with multiple frames of reference (MFR) (the coupling between rotating and stationary components).

A new LaunchPoint VAD, PediaFlow™ is intended to deliver a flow rate of 0.3 to 1.5 l/min against 100 mmHg pressure rise to neonates and infants weighing 3 to 15 kg. The PediaFlow was designed with a magnetically suspended, mixed-flow style impeller with a single annular flow gap between the rotor and housing to avoid unfavorable retrograde flow and separation. The shear stress transport (SST) model, a low Reynolds number turbulence model, was selected for the turbulent flow simulation, which was justified by the representative Reynolds number of ~30,000 based on the impeller outlet diameter and the pump tip speed. Although blood exhibits non-Newtonian behavior at very low shear rates, many studies have shown that blood can be modeled as a Newtonian flow at a shear rate larger than the threshold of a  $100 \text{ s}^{-1}$ . The

shear rate in the computational model of the PediaFlow is much larger than this threshold, so Newtonian blood with a constant viscosity of  $0.0035 \text{ Pa}\cdot\text{s}$  and a density of  $1040 \text{ kg}/\text{s}^3$  was assumed for the simulations.

The CFD-predicted velocity vectors at both the mid-span blade-to-blade region of the impeller and the vane-to-vane region of the stay-vanes show a very smooth distribution without any vortices at the nominal flow condition for the optimized PediaFlow model. As literature is replete with anecdotal evidence that recirculating flows lead to attachment of platelets to biomaterial surfaces — which in the clinical VAD setting can promote blood clot formation — reverse flows and vortices are undesirable. The CFD results found that a smooth and gradual transition in the secondary flow velocity was present at the curvature of one inflow and outflow cannula geometry. This graduation helps to prevent separation and reversal flow for the primary flow velocity. In addition, the predicted pathlines of representative particles through the entire flow region did not exhibit any vortices.

The exposure of blood elements to shear stress above a certain threshold as a function of exposure time can cause hemolysis, which actively breaks open the red blood cells; activate platelets, which can cause clotting problems; and denature proteins, which



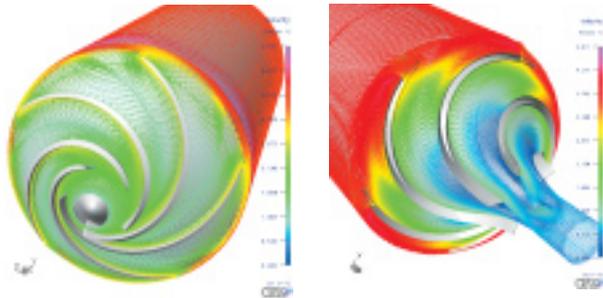
The PediaFlow ventricular assist device provides long-term cardiac support for infants.

alters the proteins so they can no longer carry out their cellular functions. Thus, it is desirable to minimize the shear stress that blood passing through the pump may experience. Using the results of the CFD simulation, a plot of shear stress versus exposure time for particles passing through the pump demonstrates relative uniformity within the annular flow gap region, but it is less uniform within both the impeller and stay-vane regions. The overall mean blood damage through the entire domain of the model is divided according to the three main regions of the flow path: impeller, annular gap and the stay-vane. The analysis reveals that the hemolysis level in the annular gap region is highest, accounting for more than 50 percent of the total, while the level of hemolysis in the impeller region and stay-vane region is almost the same, each causing approximately 20 to 25 percent of the total blood damage.

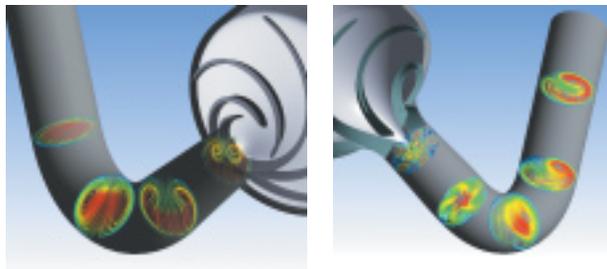
CFD-based design optimization with the integration of the ANSYS CFX solver can significantly reduce the design optimization cycle from years, compared to the traditional trial-and-error methods, to just several months. It provides detailed and useful flow field information from which blood damage may be computed, and it also predicts the hydrodynamic characteristics such as the relationship of developed pressure and efficiency to flow rate. ■

This research was supported in part by NIH Contract No. HHSN268200448192C (N01-HV-48192).

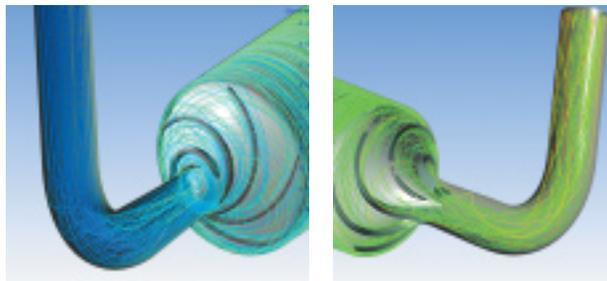
PediaFlow is a trademark of WorldHeart, Inc.



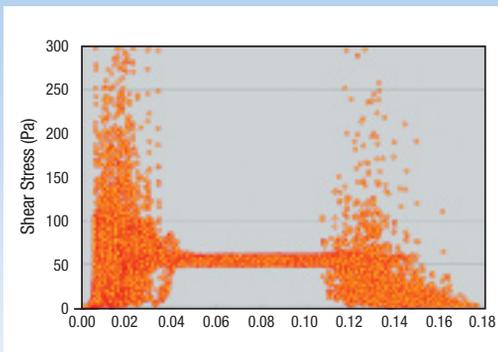
Predicted smooth velocity vectors at mid-span blade-to-blade region of the impeller (left) and mid-span vane-to-vane region of stay-vanes (right)



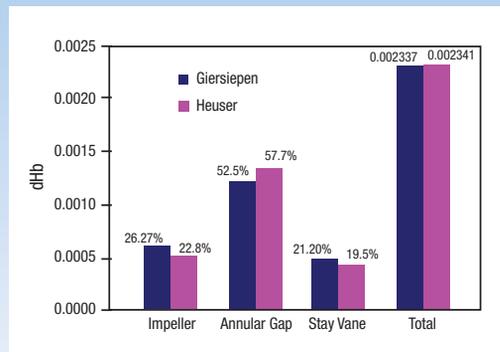
Secondary flow streamlines at sections of inflow cannula (left) and sections of outflow cannula (right)



Pathlines of particles at inflow cannula and impeller side (left) and stay-vanes side and outflow cannula (right)



Shear stress history from impeller inlet to stay-vane outlet



Proportion of total blood damage at different pump components under nominal flow condition