Researchers Keep Blood Flowing in Cardiac Pumps Designed With CFD

The Project: To improve catheter and rotary pumps used by patients waiting for a heart transplant. The Solution: Use analysis software to see inside the pump and minimize potential blood-flow problems.

Markus Lorenz is a research engineer at Cleveland Clinic's Lerner Research Institute (LRI), 9500 Euclid Ave., Cleveland, OH 44195. He is leading the R&D effort on catheter and rotary pumps. Lei Gu is a research engineer and the LRI researcher primarily responsible for the CFD analysis. Mark Goodin is director of LRI's Medical Device Innovations Group in the Biomedical Engineering Department. William A. Smith, D. Eng., P.E., is part of the associate staff. More information about catheter and rotary pumps is available by contacting Smith at 216-445-9334 or SMITHW1@ccf.org. More information on CFD software is available by contacting ANSYS Inc., Southpointe, 275 Technology Dr., Canonsburg, PA 15317 at 800-937-3321 or ansysinfo@ansys.com.

[1]
Surface streamlines on the inflow tube illustrate the complex 3D flow field. Flow patterns near the surface suggest the presence of vortices near the outer edge of the inflow.
(click the image to enlarge)

By Markus Lorenz, Lei Gu, Mark S. Goodin, and William A. Smith, D. Eng., P.E.
Researchers at the Cleveland Clinic Foundation, one of the world's largest research hospitals, are using computational fluid dynamics software to advance catheter and rotary pumps for cardiac patients.
These pumps keep blood flowing through a patient's body while he or she waits for a heart transplant. Catheter pumps are designed to assist blood flow for relatively short periods of time. They are inserted into a patient's heart through the femoral artery. They temporarily keep blood flowing during cardiology procedures and similar operations. Rotary pumps are designed for continuous flow, sometimes for months, and literally replace the blood pumping function of the heart.

Cleveland Clinic Foundation's research in artificial hearts dates back half a century when Dr. Willem J. Kolff implanted the world's first artificial heart in a dog in 1957. As a result of today's analysis work with computational fluid dynamics (CFD) software, tomorrow's artificial hearts will be both smaller and safer for patients. The analysis work lets engineers at Cleveland Clinic Foundation's Lerner Research Institute (LRI) see what happens inside the pump so that they can minimize potential blood-flow problems.

Catheter Pumps

At the pump’s center is an impeller that is 4 mm in diameter and rotates at 60,000 RPM. The rest of the pump consists of a stator, an impeller, and two sets of permanent magnets that act as bearings at the front and rear of the pump.
Blood enters through the front of the catheter pump and is forced out its sides at about a 45-degree angle. This design is best for keeping shear stresses in the pump at the lowest possible levels so that red blood cells are not destroyed. Otherwise, patients would suffer anemia and a range of blood-cell damage problems. Blood cell damage is a function of stress levels and the length of exposure to those stresses. If the exposure time is short, high stress levels can be tolerated.

Engineers had to avoid any design with local areas of slow or recirculating flows inside the pump. These would have caused blood flow to stagnate, potentially leading to life-threatening blood clots.

For these analyses, the engineering team relied on CFX-BladeGen and CFX-5 by ANSYS Inc. in Canonsburg, PA. The first step was to design an inflow stator that would result in a minimal pressure drop, which could damage blood cells, while optimizing the flow field for the rotating impeller. Multiple simulations and design iterations were performed to find an optimal stator design for the pump.

The engineering team initially chose the CFX products because of their solver’s track record of accuracy in turbomachinery analysis. As they refined their pump designs, they used CFX to help identify potential areas of stasis as well as high shear stress.

The data from the stator simulation allowed the engineers to refine the tiny impeller vanes so that the blood flows smoothly into the pump, precisely matching the angle of the vanes and minimizing harmful turbulence. Accurate three-dimensional simulations with CFX allowed engineers to set vane angles more precisely than traditional one-dimensional calculations of the flow.

[2]
Flow vectors demonstrate how blood flows through the impeller. The pump was designed to minimize shear stress levels and creation of vortices, which increase residence time of blood in the pump. Both shear stress and residence time can contribute to blood-cell destruction.
(click the image to enlarge)

To refine and optimize the pump design, LRI engineers combined prototype testing with CFD analysis. Before the analyses were run, they built a solid model of the pump in Pro/Engineer software from PTC. The 3-D CAD model was used to produce a prototype of the pump on a stereolithography machine. The prototype allowed researchers to test the performance of the initial pump design.

But the prototype tests were useful only to gauge the overall performance of the pump. The small size of the pump inhibited the use of lab instrumentation. Rather, engineers visualized the flow field inside the pump with CFX-5. Initially, they found problematic vortices downstream of the impeller vanes, which increases residence time of the blood inside the pump&#151a risk factor for clotting&#151and reduces overall performance.

The analysis results also were used to optimize the impeller design so that shear stress levels inside the pump, as well as exposure time of the blood fluids to the pump, would be minimized. Blood cells exposed to high levels of shear stress over time can be damaged or destroyed.

A densely meshed CFX model of the small device was constructed. The total length of the catheter model was about 170 mm (about 6.7 inches) but had a housing inner diameter of just 4 mm for the impeller. The final hybrid mesh was comprised of roughly 900,000 nodes and more than 3 million elements.

To obtain results that would closely correlate to the prototype results from the test stand, it was important to include geometry for the pump’s inlet and outlet section as well as for the pump itself. As opposed to finite element modeling of the structure, only the fluid regions were modeled.

Prismatic elements were inflated from the wall regions into the fluid domain in order to accurately resolve velocity gradients in the wall boundary layer. The remainder of the interior was filled with pyramidal and tetrahedral elements. The final mesh was comprised of 2.4 million tetrahedra, 850,000 wedges, and 26,000 pyramids.

The model was solved with Reynolds-Averaged Navier-Stokes (RANS) equations employing the CFX Shear Stress Transport (SST) turbulence model and solved in double precision. LRI researchers used a pair of late-model, custom-built single-processor PCs running in parallel. (Each PC has 2 GB of RAM, a 2 gigaHertz clock speed, and a 34 GB hard drive.) The operating system was Windows XP. Solving in parallel across all four CPUs, the analysis converged, or solved, within 400 iterations, requiring 20 hours of wall clock time.

[3]

(click the image to enlarge)

Rotary Pumps

New designs of the rotary pump also required CFD analysis. In a nutshell, what resulted was a simpler, less expensive, and more reliable design. Unlike devices currently on the market, LRI’s pump needs no position sensors or active-control feedback.
The pump consists of an impeller in an unusual design that contains all rotating pump parts. There is also a counter-rotating motor-stator, an inherently controlled magnetic-bearing stator, and a spiral-shaped (volute) housing. Two permanent magnets inside the impeller and six copper-wire coils in the stator combine to form a brushless DC motor to drive the pump.

To reduce the size of the pump, the inflow was designed perpendicular to the rotating axis of the pump. The inflow bends, forming an elbow. As with the catheter pump design, CFD analysis was used to ensure a minimum inflow pressure drop and minimum turbulence inside the inlet as blood flowed in. The inflow also was initially designed in Pro/Engineer.

The model was comprised of 345,000 elements with 91,000 nodes. Prismatic elements were again inflated from the wall boundaries with tetrahedral elements filling the interior mesh regions. The mesh used about 261,000 tetrahedra and approximately 84,000 pyramids.

Using the same pair of PCs, analysis took about two hours and required 100 iterations to reach convergence. In addition to the CFX solver, ANSYS CFX meshing and post-processing (CFX-Post) tools were used for pre- and post-processing of the model and results.

Currently, prototype testing on both pumps is helping determine how much hemolysis is taking place and both pumps are being readied for animal testing, which is the first step toward commercialization.
ONLINEFor additional information on the technologies discussed in this article, see *Medical Design Technology* online at www.mdtmag.com and the following websites:

&#149 Cleveland Clinic at www.clevelandclinic.org

&#149 ANSYS Inc. at www.ansys.com

&#149 PTC at www.ptc.com