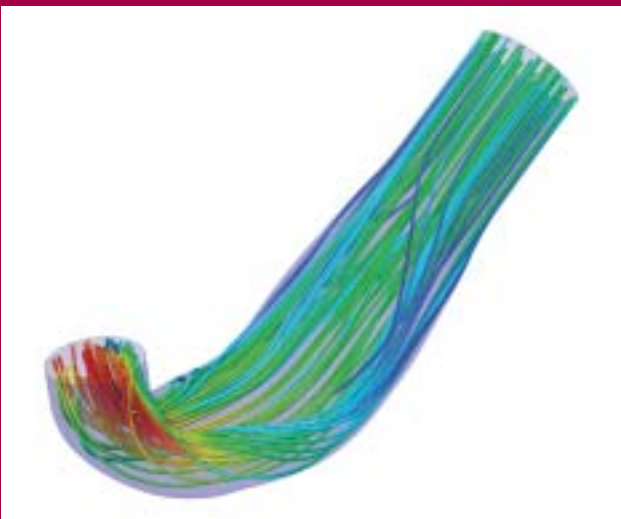


## Researchers Keep Blood Flowing with Cardiac Pumps Designed with CFD

Devices maintain blood flow for patients awaiting heart transplants or other surgical procedures.



Velocity vectors on a 2-D slice plane of the inflow tube allow engineers to visualize the flow field through the pump inlet.



Surface streamlines on the inflow tube illustrate the complex three-dimensional flow field. Flow patterns near the surface suggest the presence of vortices near the outer edge of the inflow.

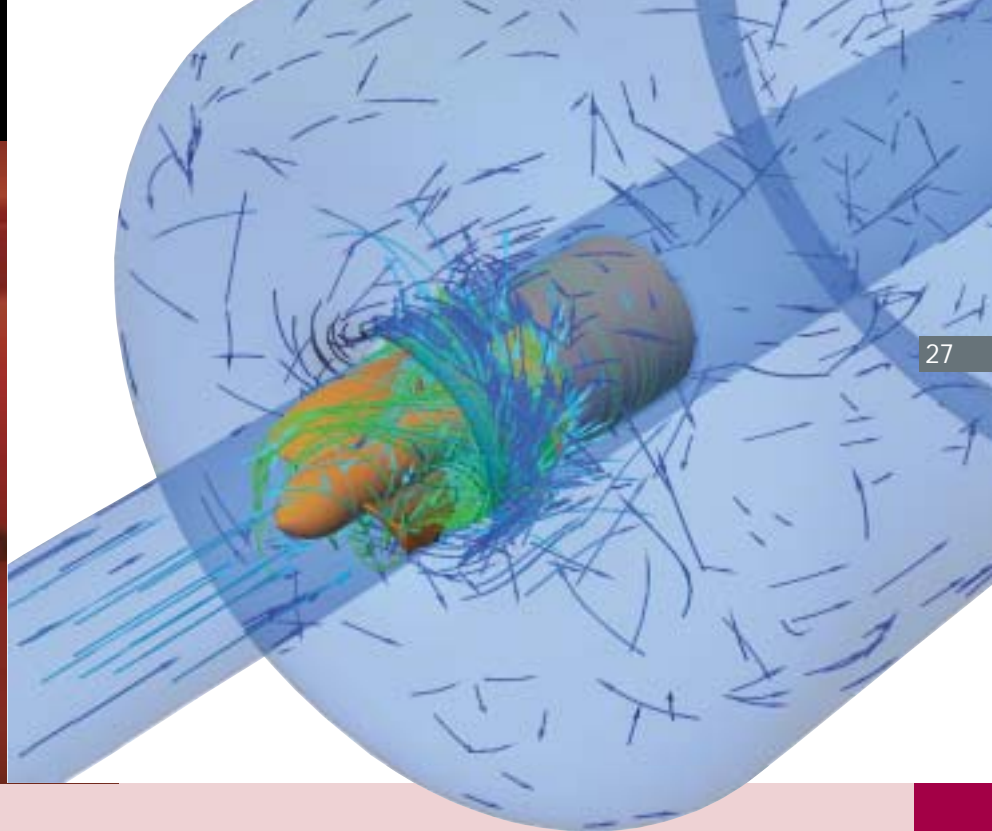
Researchers at the Cleveland Clinic Foundation (CCF), the renowned Cleveland, OH, medical institution and one of the world's largest research hospitals, are using ANSYS CFX computational fluid dynamics (CFD) software to advance the state of the medical art in catheter and rotary pumps for cardiac patients.

These pumps keep blood flowing through a patient's body while he or she waits for a heart transplant or other major surgical procedures.

- Catheter pumps are designed to assist blood flow for relatively short periods of time. Inserted into the patient's heart through the femoral artery, catheter pumps temporarily keep blood flowing during cardiology procedures and similar operations.
- Rotary pumps are designed for continuous flow, sometimes for months, and they literally replace the blood pumping function of the heart.

The Cleveland Clinic Foundation has a rich and deep tradition of research in artificial hearts dating back half a century. Dr. Willem J. Kolff implanted the world's first artificial heart there, in a dog, in 1957. As a result of today's work with CFD, tomorrow's artificial hearts will be both smaller and safer for patients. The analyses let engineers see what happens inside the pump so they minimize potential blood-flow problems, which can threaten patients' lives.

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.



### Catheter Pumps

The research and development effort on the catheter pump is being led by Principal Research Engineer Markus Lorenz of the Cleveland Clinic's Lerner Research Institute (LRI). At the pump's center is an impeller only 4 mm in diameter that 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. Lorenz said this design is best for keeping shear stresses in the pump at the lowest possible levels so that red blood cells are not destroyed. If that happens, patients suffer anemia and a range of other 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, Lorenz pointed out.

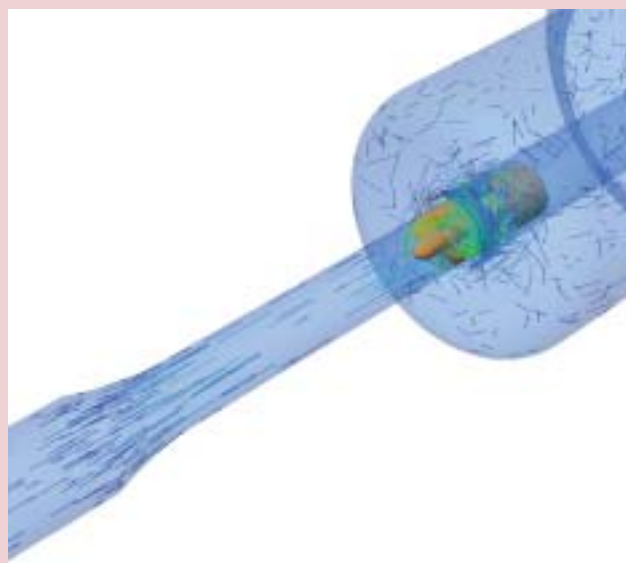
Lorenz noted that engineers had to avoid any design with local areas of slow or recirculating flows inside the pump. They cause the blood flow to stagnate, potentially leading to life-threatening blood clots.

For these analyses, Lorenz and his team relied on CFX-BladeGen and CFX-5 software from ANSYS, Inc. 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.

"We initially chose the ANSYS CFX products because of their solver's track record of accuracy in turbomachinery analysis," said Mark Goodin, former director of LRI's Medical Device Innovations Group in

the Biomedical Engineering Department. "As we refined the pump designs, we used ANSYS CFX to help identify potential areas of stasis as well as high shear stress."

The data from the stator simulation allowed LRI 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. "Without the ANSYS CFX software, it would have been difficult to precisely set vane angle because the engineers would have to rely on simple one-dimensional calculations of the flow field," Lorenz said.



Velocity vectors illustrate the flow of blood from the inflow into the impeller. It is important to design the heart pump so that blood does not stagnate as it flows through the pump.

"As we refined the pump designs, we used CFX to help identify potential areas of stasis as well as high shear stress."

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 from Parametric Technology Corp. 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 really useful only to gauge the overall performance of the pump. "With CFX-5, we were able to visualize the flow field inside the pump," said Lorenz. "We initially found that we had vortices downstream of the impeller vanes. That was a problem because it increases residence time of the blood inside the pump [a risk factor for clotting] and 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," Lorenz added. "Blood cells exposed to high levels of shear stress over time can be damaged or destroyed."

Dr. Lei Gu, the LRI researcher primarily responsible for the CFD analysis, built an ANSYS CFX

model that was fairly small yet densely meshed. The total length of the catheter model was only about 170 mm (about 6.67 inches), but with a housing inner diameter of just 4 mm for the impeller. The final hybrid mesh was composed 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.

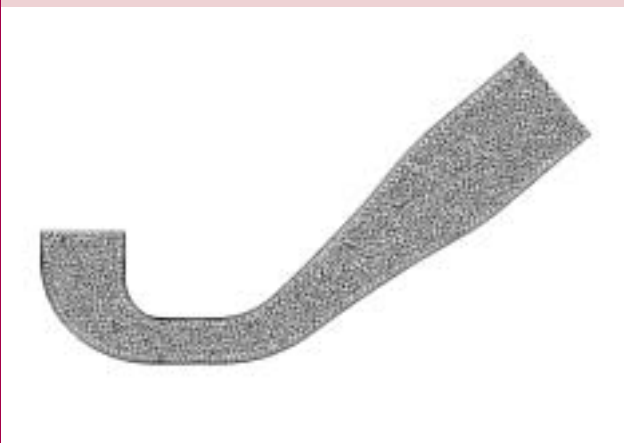
Prismatic elements were inflated from the wall regions into the fluid domain in order to accurately resolve velocity gradients in the wall boundary layer. The remainder of the interior was filled with pyramidal and tetrahedral elements. The final mesh was composed of 2.4 million tetrahedra, 850,000 wedges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

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

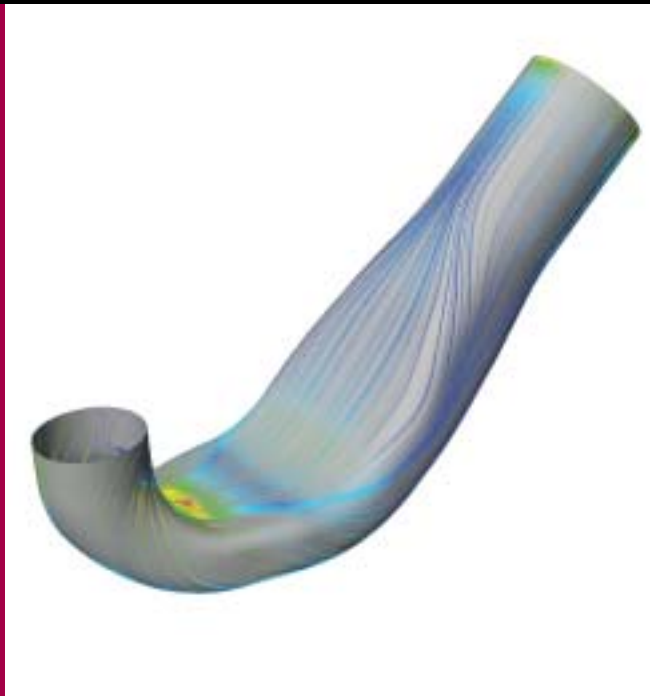
## Rotary Pumps

New designs of the rotary, or longer-term ventricular-assist, 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 needs no position sensors or active-control feedback.

The pump itself consists of an impeller, which in an unusual design contains all the rotating parts of the pump. There is also a motor-stator, an inherently controlled magnetic-bearing stator and a



A line plot of the intersection between element edges and a slice plane shows prism layers inflated from the surface mesh to allow high resolution of gradients near the wall of the inflow tube. The higher resolution gradients are required for CFX-5 to accurately predict flow through the tube.



Surface streamlines illustrate the complex three-dimensional flow field through the inflow tube. Flow patterns near the surface suggest the presence of vortices near the outer edges of the tube.



Velocity vectors illustrate the flow field in three dimensions.

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 in the catheter pump, CFD analysis was used to ensure a minimum inflow pressure drop and minimum turbulence inside the inlet as blood flowed in. Also, as with the catheter pump, the inflow was initially designed in Pro/Engineer.

The model was composed 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.

On 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.

"In the future, LRI researchers would like to analyze a model of the blood itself as it flows inside the pump. Such analysis would help determine how much hemolysis [separation of the red blood cells] actually takes place as the pump operates," said Lorenz.

To do this analysis, engineers would have to introduce imaginary particles representing the many constituents of blood, and to track what happens to each of them as they move through the pump.

As of late 2004, neither the catheter pump nor the long-term-assist (rotary) pump had yet made it to market, let alone into cardiac operating rooms. Cleveland Clinic officials report, however, that commercial partners are interested in both projects.

Currently, prototype testing on both pumps is helping to determine how much hemolysis is taking place, and both pumps are being readied for animal testing, which is the first step toward commercialization. Once animal tests are complete, the researchers expect partners to come forward to fund commercialization of the projects. In fact, Lorenz previously has worked in Germany on similar pumps that have been commercialized.

"CFX is very helpful for us and adds another layer of technical depth to the research we do," said Goodin. "Without CFD, researchers would normally make something they think would work. Then they would have to iterate experimentally to refine their designs and achieve the flow conditions that they want, an often slow, costly and painstaking process. Using CFD provides them with valuable insights to refine the designs and truly understand the flow fields within those designs." ■