# Optimizing Existing Designs through Cost-Effective Simulation

Glenn Dorsch, P.E., and Kent Keeran, Vaughan Company Inc.

**How CFD simulation provides pump manufacturers with a virtual test facility.**

The chopper pump is a centrifugal pump that utilizes a chopping action between the impeller and suction plate. These pumps were originally designed in the 1960s for use in the local dairy industry to transport manure to and from storage tanks.

Since then this design has been continually refined to earn wide acceptance for many applications which require solids handling. These pumps are used in various phases of municipal and industrial sewage treatment, food processing, and pulp and paper, where the pumped liquid contains solids that need to pass through the pump without clogging or plugging.

In centrifugal pumps which utilize a chopping action between the impeller and suction plate, the main impeller vanes extend all the way to the center hub of the impeller, and the suction plate includes two stationary fingers which protrude to the center of the suction opening. As the main vanes pass by the stationary fingers a chopping action results which macerates any solids entering the pump.

The benefit of a pump with chopping action between the impeller and suction plate over a typical non-clog or slurry pump is that it reduces the solids size of material passing through the pump. The unique chopping requirements and suction arrangement of these pumps make it difficult to apply standard impeller design practices to achieve the desired hydraulic performance.

Past practices have approximated head and flow at best efficiency for new designs, but it has been difficult to predict

A submersible centrifugal pump sits in the bottom of a pit filled with water and debris and pumps the debris-laden water to downstream equipment.

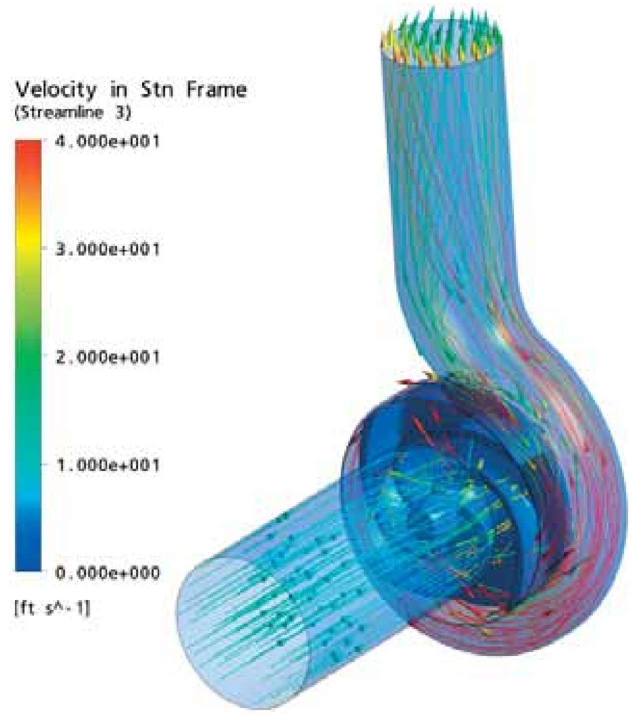
head for other flow points, and almost impossible to predict pump efficiency. And as energy costs continue to rise, developing more efficient pumps becomes increasingly critical for all pump manufacturers.

As a result, computational fluid dynamics (CFD) software, such as the ANSYS CFX system we use, has been utilized for several years to simulate pump designs. For example, consider nozzle-based mixing systems for tanks. CFD software not only has the ability to optimize mixing designs, but also to show customers what is actually happening inside the tank. After successfully simulating hundreds of mixing systems, CFD software that has a multiple frames of reference capability can be used to assist in improving pump efficiency.

**Geometry of a typical casing, impeller, and cutter bar assembly for a centrifugal pump with a chopping action between the impeller and the suction plate.**

For example, we have used Pro/ENGINEER® software for pump design for the last ten years, meaning 3D solid models for a number of the different pump designs are already available. Using this CAD modeling software, it is relatively easy to determine the fluid domain from the solid model, including volumes upstream from the impeller suction and downstream from the casing discharge.

The pump geometry is then imported into the ANSYS DesignModeler. The impeller domain and casing domain are meshed separately and assembled within the CFX pre-processor where boundary conditions are applied. The CFX solver performs the required calculations. Then results are viewed and pump performance calculated in the CFD post-processor. The ANSYS Workbench platform facilitates the entire simulation process, from geometry import through visualization.



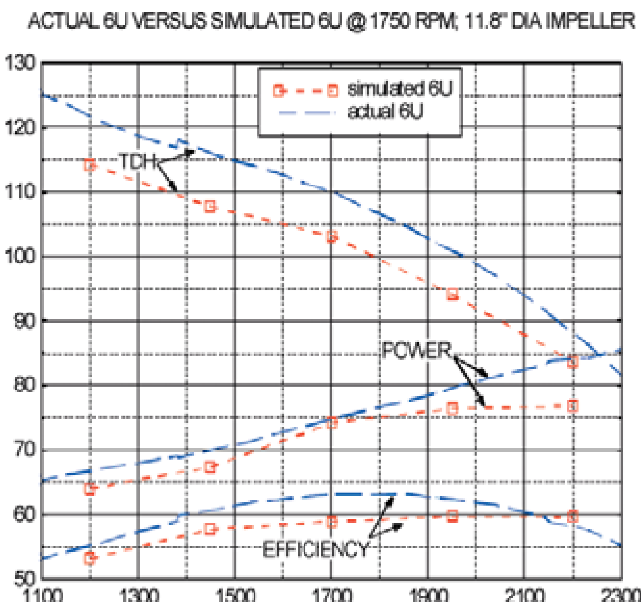

**Results of a simulation are visualized using the CFD post-processor.**

The results of simulation have been very rewarding, especially when compared with fabricating and testing prototypes. Fabricating impellers by hand, or machining them from a billet on a 5-axis CNC machine tool, is very expensive and time-consuming. A sand-casting pattern can be built for about the same cost as fabricating or machining a prototype.

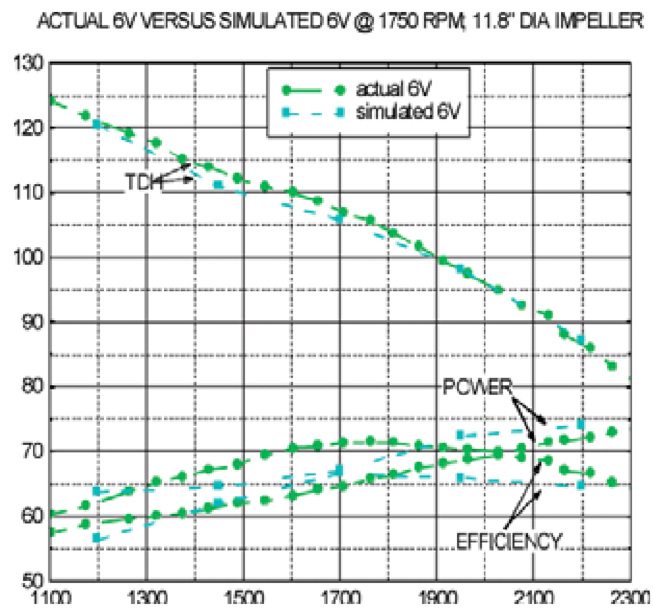
While rapid prototyping is about half the cost of fabricating or machining a prototype, it is still much more expensive than the cost of building the virtual model and running the simulation. In fact, such good performance testing correlation has been achieved between the simulation and the cast impellers created from the same design that physical prototypes are no longer required. All research and development prototypes are modeled, simulated and optimized, then go straight to production castings.

With regard to pump design, this is an ideal way to improve the efficiency of existing designs, as opposed to generating new models. A simulation is run for a given pump and the results of the simulation are compared to real performance results.

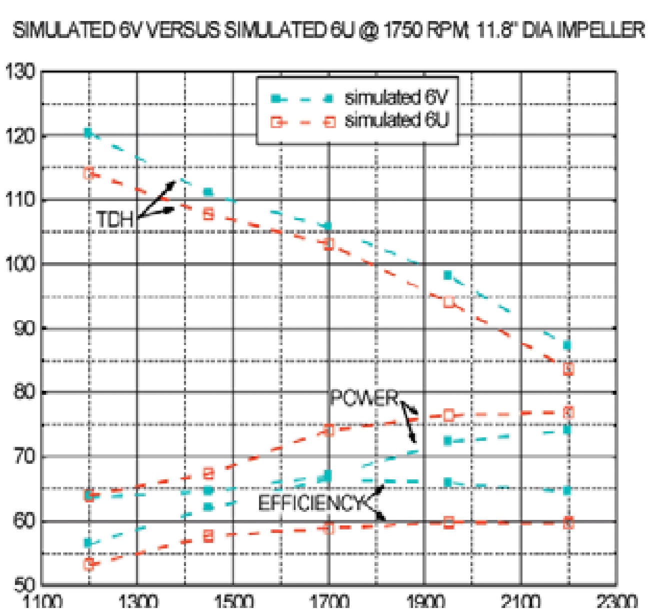
In testing the real pump, a valve on the discharge side of the pump is progressively opened or closed. At each different performance point (i.e. valve position) pressure and flow data are collected. These data points can then be connected to show total dynamic head (TDH, measured in feet), power (measured in horsepower), and efficiency (percentage) with respect




**Performance curve for a recently redesigned 6-in pump. The simulation slightly under-predicts TDH in this case because the geometry for the impeller and casing had to be reverse engineered, and there were likely some differences between the model and the actual parts.**




**Comparison of the simulated redesigned impeller with the actual redesigned impeller test results show the TDH curves match very well. Efficiency exceeds expectations, probably because very conservative simulations were run which slightly over-predicted the power required.**




**Comparison between the simulated existing impeller and the simulated redesigned impeller ensures that the redesigned impeller has TDH characteristics which are as good as the original impeller. Because the cost of virtual performance testing via simulation is very low, it is possible to try many subtle variations of impeller blade shape in order to determine the optimum combination for this particular pump design. Approximately an 8-point increase in efficiency over most of the flow range is achieved in the new design.**

to flow (gallons per minute, gpm).

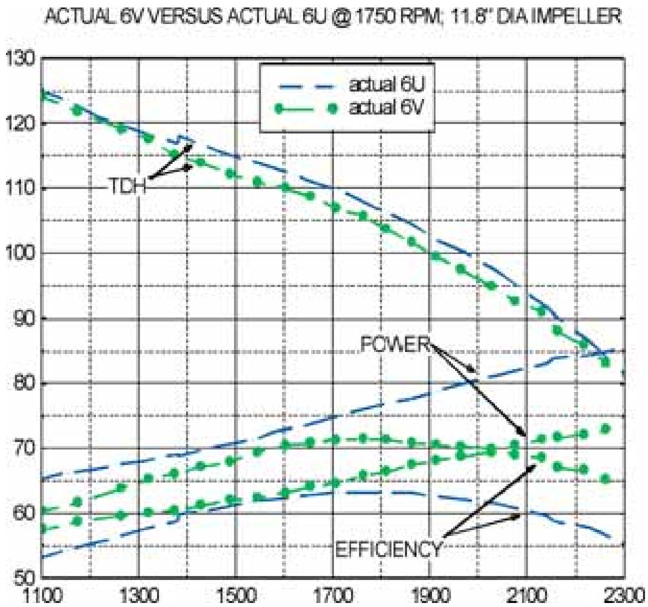
The simulation is run in a similar manner. Several simulations, at various flow rates, are performed on a given model. The performance data can then be extracted via the CFD post-processor and a virtual performance curve can be constructed for that model.

A new impeller design is then modeled, performance simulated, and compared to the simulation for the existing impeller. Originally, the impeller blade shapes were generated in Pro/ENGINEER® but the process was cumbersome and ANSYS BladeModeler blade design software is now used. This software allows for easy generation of blade shapes to meet specifications, and for the export of control curves to Pro/ENGINEER® where the solid model is constructed.

A new pattern is then built directly from the Pro/ENGINEER® model for accuracy, and the performance of the actual redesigned impeller is then physically tested at the test facility.

The simulated pump performance accurately predicts the actual pump performance in all seven models completed to date. Similar results are expected for the two models currently being redesigned.

Pump manufacturers can utilize CFD simulation software, in combination with 3D CAD software, to effectively build and test prototype pump models at an engineer's desk. The relatively low cost of this type of research and development program allows testing of a larger number of impeller blade shape variations than would be possible otherwise. This




**When comparing performance of the original pump and the new design, the TDH is a very close match and an 8-point to-9-point improvement in efficiency is achieved across the entire flow range. The simulated pump performance has accurately predicted the actual pump performance in all seven models completed to date.**

approach enables better optimization of any given design.

In addition, it is a simple matter to extract a wide variety of information, including not only pressures and flows but also component forces, to better optimize the complete pump design. This optimization includes not only hydraulic design but also mechanical design, such as the bearing selection via accurate impeller loads.

CFD software has proven to be a tool that can accurately and efficiently help optimize designs through cost-effective simulation.

P & S

*Glenn Dorsch, P.E., is the vice president of engineering and Kent Keeran is a senior engineer at Vaughan Company Inc., 364 Monte Elma Road, Montesano, WA 98563, 360-249-4042, Fax: 360-249-6155, www.chopperpumps.com.*

*ANSYS, Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, 724-746-3304, Fax: 724-514-9494, ansysinfo@ansys.com, www.ansys.com.*