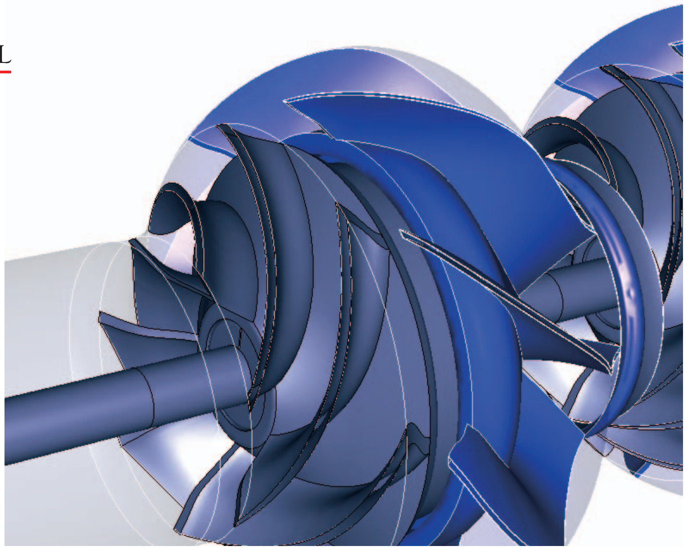
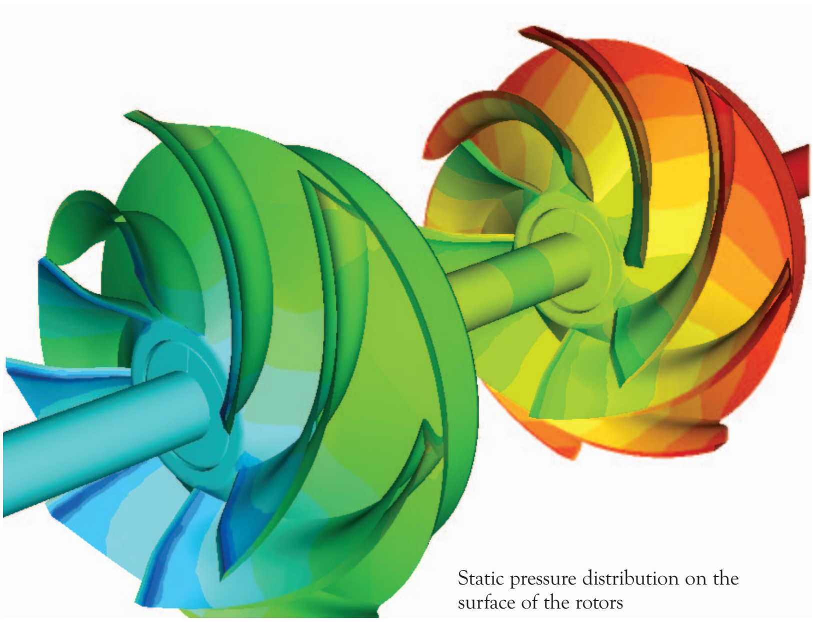


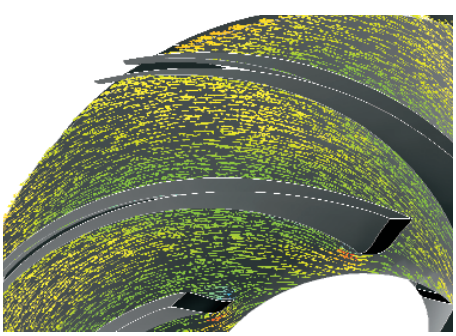
An isometric view of the pump

Staging a Better Pump Design

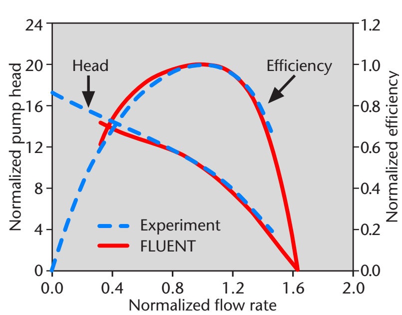
By Onur Konuralp, Layne Bowler Vertical Turbine Pump Company Inc., Ankara, Turkey; and Ahmet Acikgoz and Emre Ozturk, ANOVA Ltd., Atasehir-Istanbul, Turkey



Static pressure distribution on the surface of the rotors



Relative velocity vectors on a plane between the rotor blades



Comparison of pump performance and efficiency with data

MIXED FLOW PUMPS are primarily axial flow pumps, but they impart some degree of radial and swirling momentum to the pump fluid as it passes through the rotor section. They are popular for pumping water in tight spaces, so are used for residential wells, municipal water works, industrial applications, and even for powering small water craft. Stationary vanes are sometimes used to straighten the flow downstream of the rotors. When two or more pump stages are employed, the pressure rise across the pump (head) can be increased without a corresponding increase in the pump flow rate (capacity).

A recent simulation focused on two stages of a vertical turbine mixed flow pump. Each of the two identical stages of the pump consists of a rotor and a stator. In-line inlet and exit sections are used upstream and downstream of the pump components. A tetrahedral volume mesh was used for the simulation because it would have been much more time-consuming to build a hexahedral mesh for such a complex geometry. During the creation of the volume mesh, a high mesh density in the vicinity of the rotor blades was sought. GAMBIT's Size Function tool was used to smoothly vary the mesh density from the rotor blade region to nearby regions where the mesh demands were not as great. This approach resulted in an initial tetrahedral volume mesh of approximately 2.2 million elements. The rotation of the rotors was simulated using the steady-state multiple reference frames (MRF) model in FLUENT.

The standard k-ε turbulence model with non-equilibrium wall functions was used for the calculation in which water was used as the working fluid. Pressure boundary conditions at the flow inlet and outlet were used to prescribe the head across the pump. After a partial solution was obtained, the grid was adapted based on the y^+ values. This resulted in a final element count of about 2.6 million cells. The calculation continued, and the mass flow rate and efficiency of the pump were predicted by the final converged CFD solution. The solution also demonstrated the pressure rise – radially and axially – in the pump.

The relative velocity vectors inside the rotor section, viewed from the rotating frame of the rotor, showed that the water flows smoothly between the rotor vanes. Conserving a regular flow pattern in this region increases the efficiency of the pump. In the stator region, however, an irregular velocity distribution suggested that the shape of the vanes and hub surface could be improved. In the future, optimization of the stator vanes is planned to improve the pump performance.

Pump performance and efficiency curves can be obtained by changing the pressure at the pump outlet (and therefore the pump head) and recomputing the mass flow rate through the pump. Several different pressure outlet values were tested, and were found to be within 3% of experimental measurements. ■

More.info@
www.laynebowler.com.tr