RISING TIDE

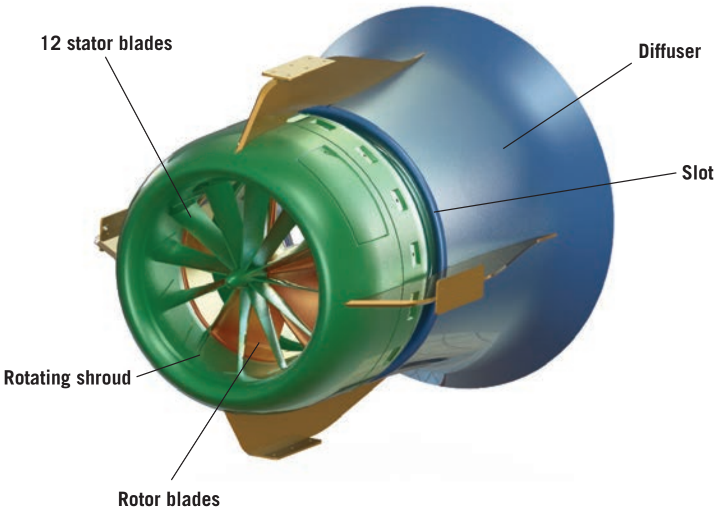
By employing simulation, a consulting firm optimizes the design of an innovative tidal current power generator to produce four times as much power as earlier designs.

By Richard Billett, Senior Consultant, Gilmore Engineers Pty Ltd, Brisbane, Australia

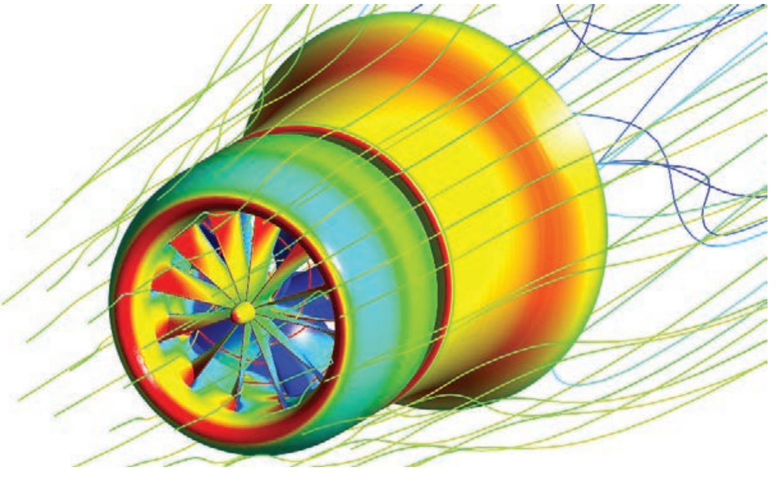
Addressing growing worldwide electricity consumption and the desire to reduce greenhouse gas emission from fossil-fuel–based resources drives the demand for renewable energy technologies. Along with harnessing wind and solar power, capturing the vast kinetic power in the world's tidal currents, ocean streams and river flows is one of the most promising sources of renewable energy. Predictability of tidal currents and ocean streams means that underwater power stations could form part of the baseload power supply that produces energy at a constant rate. This offers a significant advantage over other unpredictable or intermittent renewable energy alternatives. An earlier generation of experimental marine power generators used a shrouded propeller with blades that were supported only where they connect to the shaft. However, resulting stresses on the blades led to failures, and the search for solutions has resulted in the use of expensive materials.

Inventor Michael Urch's design avoids the failure problem by connecting the outer diameter of the rotor blades to the shroud. The rotating shroud connected to the rotor, combined with a stationary shroud that expands the effective flow area, provides excellent flow guidance when compared to an open turbine. This design increases efficiency and power output. A circumferential slot on the interior helps to maintain flow attachment to the walls in the portion of the shroud downstream of the rotor (where the shroud acts as a diffusor). By avoiding flow separation, drag decreases and the amount of power produced increases. A stator at the turbine inlet introduces a pre-swirl to the flow so the rotor can extract more power. Gilmore Engineers

Complete design optimization took 4 percent of the time that would have been required to optimize the design using the build-and-test method and 25 percent of the conventional CFD approach.



▲ CAD model of marine power generator



▲ CFD results show pressure plotted on the surface of the power generator with velocity streamlines.

▲ Prototype of SeaUrchin marine power generator

was contracted to evaluate the concept, accelerate design assessment and optimize the design. Engineers employed ANSYS computational fluid dynamics (CFD) simulation, which made it possible to considerably improve the output and significantly reduce development time over standard build-and-test methods.

TRIAL AND ERROR METHOD

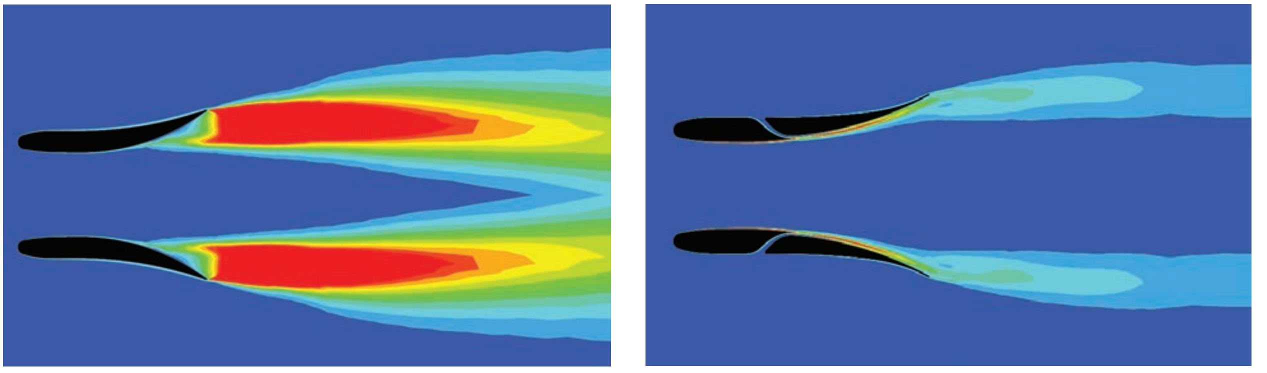
Urch basically developed this unique design concept on a napkin, but realized that it was essential to optimize the design, validate the concept and provide an estimate of how much power it would generate before taking the next step. The traditional approach would have been to build a series of prototypes to evaluate the performance of different shroud and blade designs. Engineers would have set up operations in a water flow facility to run a series of tests. During these tests, flow and pressure could have been measured only at a few discrete points, which would have restricted the data obtained from each test. The complexity of experimental setup and the limited insight gained would have slowed the development process so much that it would have taken about one year to iterate to an optimized design.

SIMULATION APPROACH

Elemental Energy Technologies Ltd. was formed to commercialize the design concept, called the SeaUrchin™ marine power generator. This company contracted with Gilmore Engineers, an engineering consulting firm, to optimize the challenging design using simulation. The shroud design was complicated because engineers needed to accurately simulate the boundary layer as water moved from inlet to outlet to identify any tendency of the flow to separate. Separation has a major impact on turbine performance. The complexity of flow patterns in the boundary layer required a fine mesh for resolution of turbulent flow in this area. The other challenge was to take into account the motion of the blades through water.

One approach to analyzing the turbine would have been to use a full 360-degree transient simulation to model the motion of the blades through the water. Turbulent

ENHANCING DESIGN PRODUCTIVITY WITH ANSYS TURBOMACHINERY SOLUTIONS
ansys.com/93tide



▲ Turbulent kinetic energy plotted over this shroud design shows that flow separates from the diffusor. Inefficient power generation (left) and efficient power generation (right).

flow at the boundary layer would be modeled with a Reynolds-averaged Navier–Stokes (RANS) turbulence model. Using a model with this level of complexity would have taken approximately as long as the build-and-test method for each iteration. However, the model would have provided much more information than physical testing, including flow velocity and pressure at any point in the computational domain, so engineers estimated that they could have optimized the design in three months using this approach.

ROTATING AND TURBULENCE MODELS KEY TO SOLUTION

Gilmore engineers looked for an even faster method to optimize the design. They selected ANSYS CFX CFD software because it provides models and infrastructure for accurate, robust and efficient modeling of rotating machinery. Since the marine power generator possesses rotary symmetry, they used a five-degree periodic model to conserve computational resources. To represent the blades, they began with a simplified model with a porous region that takes energy out of the flow. This model had between 100,000 and 200,000 elements. Engineers used the shear stress transport (SST) turbulence model, which is as economical as the k-ε model but offers much higher fidelity, especially for separated flows, to provide answers on a wide range of flows and near-wall mesh conditions.

This model could be solved very quickly even on a desktop personal computer, which was what engineers were using at the time. They simulated about 30 sizes and shapes of the shroud in the course of a week, focusing on the diffusor or draft tube region, to determine the design that provided the lowest pressure while maintaining flow attachment to the wall. Further analysis was then performed on the best-performing shapes by varying the size, shape and number of slots and taking into account production costs. Over the course of these iterations, they increased the expansion rate of the turbine by 25 percent.

ITERATING TO AN OPTIMIZED DESIGN

With the shroud optimized, the blades were solved in a rotating reference frame, and engineers used the frozen rotor model to connect the rotating component to the stationary components. They performed computations in a steady-state mode, based on the assumption of quasi-steady flow around the rotating component at every rotation angle. The model size was increased to about 10 million elements. Engineers first simulated an early physical prototype to validate the simulation model. The physical prototype generated 1,484 watts with a coefficient of power (Cp) of 0.46. Cp is the electricity produced divided by the total energy available in the water. In this case, the simulation model predicted a power generation of 1,600 watts and a Cp of 0.50, which was very close to the experimental results considering the difficulty of accurately matching the physical test setup.

Next, engineers ran a series of 10 more iterations on the blades using the optimized shroud design. They increased the amount of torque generated by the turbine by 15 percent compared to the initial design. The design optimized by CFD generated 3,892 watts with a Cp of 1.22, an improvement of nearly 150 percent over the initial design. The Cp exceeds 1.0 because it is calculated based on the inlet area, while the outlet area is almost four times as large. The complete design optimization process took about two weeks, 4 percent of the time that would have been required to optimize the design using the build-and-test method and 25 percent of the conventional CFD approach. The SeaUrchin recently won first place in the annual Engineering Excellence Awards sponsored by the Newcastle Division of Engineers Australia and The Australian Awards.

Gilmore Engineers Pty Ltd is supported by ANSYS channel partner LEAP Australia Pty Ltd.

The SeaUrchin recently won first place in the annual Engineering Excellence Awards sponsored by the Newcastle Division of Engineers Australia and The Australian Awards.