ANSYS

Excellence in Engineering Simulation

ADVANTAGE

SPECIAL ISSUE 2016

Best of

Turbomachinery

Pushing the Envelope

Shaking All Over

Pumped

Table of Contents

Realize Your Product Promise®
If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge, or put on wearable technology, chances are you've used a product where ANSYS software played a critical role in its creation.

ANSYS is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

Join the simulation conversation
ansys.com/Social@ANSYS

Contact Us
Web: www.ansys.com
Phone: 844.Go.ANSYS (844.462.6797)
Email: info@ansys.com

EDITORIAL

AS EQUIPMENT ROTATES, SO GOES THE WORLD

By **Brad Hutchinson**, Global Industry Director, Industrial Equipment & Rotating Machinery, ANSYS

Turbomachinery applications play a significant role in most industries: power generation, oil and gas, aeronautics, HVAC, chemical processing, healthcare and automotive. Efficiency gains can result in global improvements, particularly in the carbon footprint. While stringent emissions standards drive industrial equipment design, other factors play important roles. Fuel prices are volatile. Emerging OEMs create pockets of increased competition with resulting shortened time frames. Offshore, deep-water and shale gas reservoirs call for dramatic technology advances. Equipment is pushed to new operational boundaries, yet durability demands remain constant. Turbomachinery design has become a series of trade-offs.

Not surprisingly, rotating equipment companies increasingly leverage engineering simulation to build in efficiency as they reduce emissions, fuel burn and time to market. To achieve targets, developers look at improving all aspects of machine performance. Industry leaders develop and use high-temperature tolerant and lightweight materials to enable higher firing temperatures and reduce weight, while maintaining or improving machine reliability. Virtual design is especially valuable in incorporating smart capabilities that control fuel burn, automate load switching, monitor remote equipment and gather data. Advanced thermal simulation is particularly important as temperatures increase and parts are subjected to increased cycling as a result of more varied operational requirements. High-performance computing now enables simulation with greater fidelity and realism, and provides engineers with rapid turnaround and increased throughput required for digital optimization. Engineers can refine designs significantly before physical tests are performed, accelerating the process and reducing risk.

This "best of turbomachinery" issue of *ANSYS Advantage* includes many examples that validate the value of engineering simulation in building better products.

COMPRESSION AND GAS MOVING

Large centrifugal compressors used in the oil and gas industry must meet several stringent requirements, including high efficiency, high pressure, high reliability and small footprint. Recent emphasis on wide operating ranges increases the challenge. Dresser-Rand leveraged simulation in designing compressor stages to operate at higher flow coefficients and inlet-relative Mach numbers. The result was nearly a 10 percent improvement in the surge margin.

Efficient equipment reduces power consumption as well as operating costs. Rotating machinery company Continental Industrie designed a centrifugal compressor with a potential for up to 5 percent energy savings during wastewater treatment operations — an annual savings of $6,000 to $20,000 per compressor. Using simulation, the company completed the design with a three-person team in a reduced time frame, and still the first prototype met performance requirements. The aggregate savings would be substantial if even one compressor was adopted by each of the 20,000 U.S. municipal wastewater treatment plants.

In the automotive sector, turbochargers are used to get more power out of smaller engines with the additional objective of not affecting a driver's perception of handling and performance. The ideal compressor (the heart of a turbocharger) is efficient over a broad operating range and has low inertia while complying with package size limitations, robustness and cost constraints. PCA Engineers easily meets these complex requirements by using a highly automated simulation process that provides high-fidelity aerodynamic, structural and thermal information, reducing engineering effort and time to market.

HYDRAULIC TURBINES

Today's hydraulic equipment must cover an increased operating range and cycle more frequently. In the hydropower industry, plants experience enormous demand swings. Engineers at ANDRITZ HYDRO analyze turbine performance over time, under conditions that are constantly changing. To quickly retrofit aging plants and troubleshoot performance, they successfully apply virtual analysis to replicate the incredible level of product detail.

Strong vibration and pressure pulsation in hydraulic turbomachinery may be harmful to machine performance, longevity and safety. Voith Hydro observed strong vibrations that can cause fatigue cracking in the guide vanes of a large Francis water turbine. It leveraged multiphysics simulation to solve a very difficult field problem.

Grundfos, which develops circulator pumps for HVAC, uses simulation optimization to investigate hundreds of designs without manual intervention. The process reduced overall design time for a new pump by 30 percent and saved approximately $400,000 in physical prototyping.

THERMAL TURBO

Turbines must run at very high temperatures to reduce fuel burn, but they require internal cooling to maintain structural integrity and meet service-life requirements. Engineers used simulation to evaluate state-of-the-art turbine-blade cooling-channel geometries and developed an innovative geometry that outperforms existing designs.

Simulation delivers robust design with a high degree of confidence that a product will operate as expected. Pratt & Whitney leverages simulation from the earliest stages of aircraft engine design to improve both development speed and product fidelity.

Turbomeca used embedded software simulation on its helicopter engine control system, decreasing development time by 30 percent. The process also reduced coding errors.

Turbomachinery design presents many challenges for R&D teams. Analyzing flow requires high-fidelity simulation tools, and virtual analysis is dependent on compute power. As technology advances into the future, expect to realize more high-fidelity solutions, and more detailed design of experiments (with thousands of fast runs) to optimize components. Nano- and mega-scaled grids will together provide accurate resolution for even the most challenging problems. Multiple physics analysis — including aero, mechanical, thermal and vibration — will be part of every design handbook, used in parallel, interactively and simultaneously. Engineers then will be able to develop turbomachinery that incorporates the right blend of trade-offs.

Pushing the Envelope

CFD simulation contributes to increasing the operating envelope of a centrifugal compressor stage.

By James M. Sorokes, Principal Engineer; **Jorge E. Pacheco**, Manager, Aero/Thermo Design Engineering; and **Kalyan C. Malnedi**, Manager, Solid Mechanics Group, Dresser-Rand Company, Olean, U.S.A.

Centrifugal compressors, also called radial compressors, play a critical role in many process industries, including oil and gas, petrochemical, and gas transmission. These machines are used to compress a gas or a gas–liquid mixture into a smaller volume while increasing its pressure and temperature.

COMPRESSOR DESIGN CHALLENGES

Process industries are looking for smaller-footprint compressors for space-sensitive applications, such as offshore, subsea and compact plant designs. Dresser-Rand reduces compressor footprints by designing stages to operate at higher flow coefficients and higher machine or inlet-relative Mach numbers. The company is among the largest global suppliers of rotating equipment solutions for long-life, critical applications.

In recent years, the industry has placed greater emphasis on achieving a wide operating range so that, for example, compressors can handle a wider range of flow rates at different stages of a well's lifecycle. Engineering simulation is an important tool in addressing these market challenges. Dresser-Rand has been using ANSYS CFX software since the 1990s to develop many new compressor designs for process industries and other applications.

Two factors limit the overall operating range of a compressor: surge or stall margin, and overload capacity. Surge or stall margin limits the compressor's ability to operate at flow rates lower than design, while overload capacity limits the ability to operate at higher rates. Rotating stall arises when small regions of low momentum or low pressure (referred to as stall cells) form in the flow passages and begin to rotate around the circumference of the compressor. These flow and/or pressure

Dresser-Rand designs compressor stages to operate at higher flow coefficients and higher machine or inlet-relative Mach numbers.

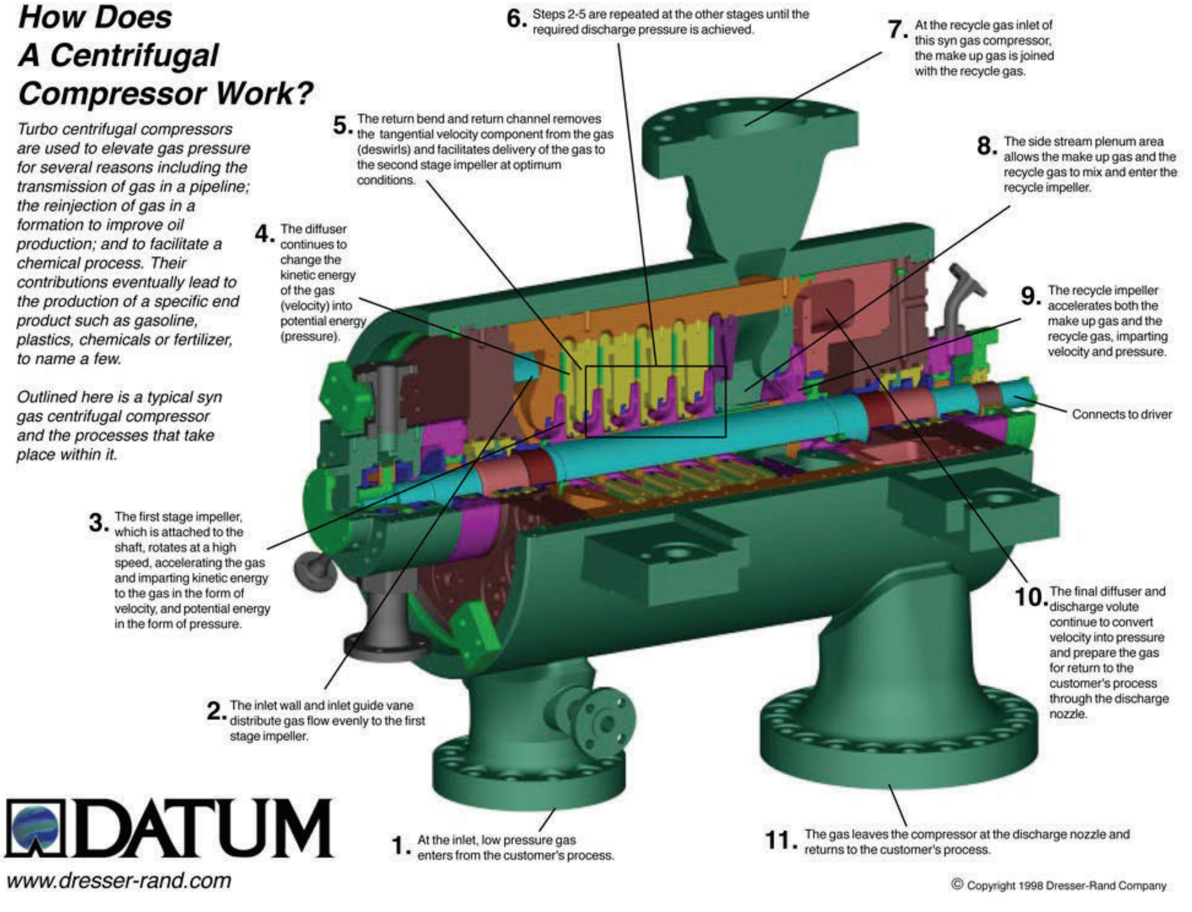
disturbances cause unbalanced forces on the compressor rotor, leading to unwanted vibration issues and reduced compressor performance. Surge occurs when the compressor is no longer able to overcome the pressure in the downstream piping and pressure vessels, and the flow is forced backward through the compressor.

For most centrifugal stages that operate at high inlet-relative Mach numbers, low-momentum regions can form along the shroud side of parallel-wall vaneless diffusers. Typically the size of this region increases as flow is reduced until diffuser stall results. In developing a new high-head stage for a high–Mach number compressor, the Dresser-Rand team observed an interesting phenomenon both in computational fluid dynamics (CFD) simulation and test results: A sudden migration of the low-momentum region from the shroud side to the hub side of the diffuser occurred as the flow rate reduced, just prior to stall [1]. The impeller used in the study is operated over a machine Mach number range of 0.85 to 1.20. The initial design had a vaneless diffuser that was pinched at the shroud and then followed by a parallel wall section. In analyzing test results, engineers established that the shift of the high-momentum region occurred much earlier for this high-head stage than for lower-head stages. As a result, the surge margin was significantly lower than low-head stages, an unacceptable drop in operating range. Since the stationary components were stalling before the impeller due to low momentum shift, the team decided to use CFD to optimize the diffuser and return channel.

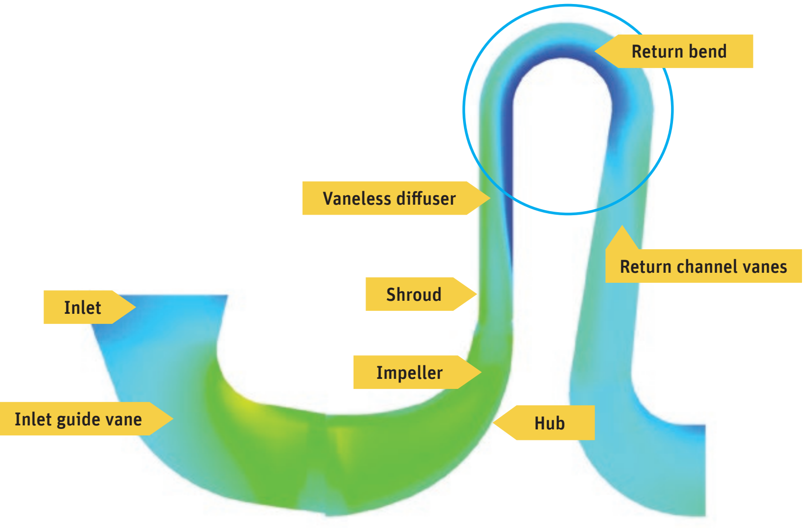
USING CFD TO OPTIMIZE THE DESIGN

Dresser-Rand engineers conducted all analyses using ANSYS CFX software for a sector model that included the upstream inlet guide vane, impeller, diffuser, return bend, return channel and exit section. In this case, the grid was composed of more than 5 million total elements using a tetrahedral mesh with wedge elements for the boundary layers. Engineers modeled the interfaces between stationary and rotating components using a stage interface that performs a circumferential averaging of the fluxes through bands on the interface. The k-epsilon turbulence model and a high-resolution discretization scheme were used.

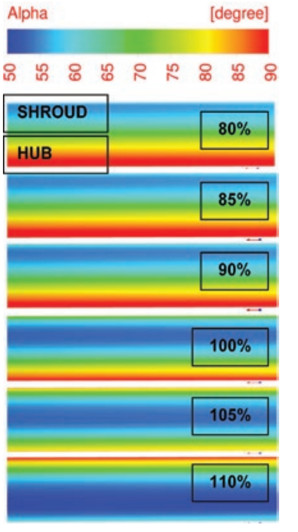
The team evaluated several combinations of pinch, shroud-tapered and



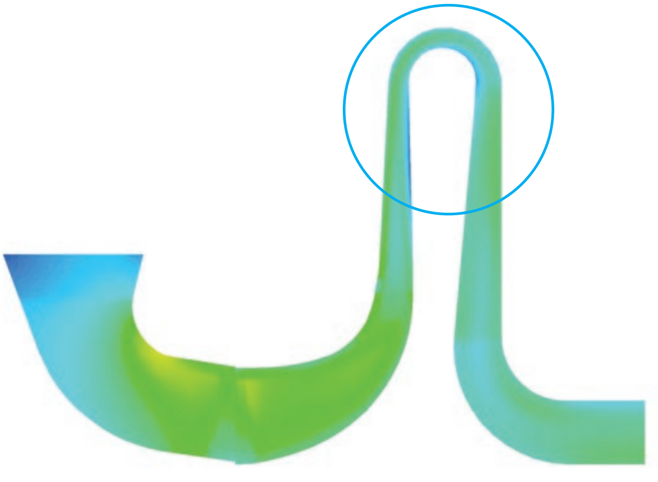
▲ **Centrifugal compressors operate by adding velocity pressure or kinetic energy to the fluid stream and then converting that kinetic energy into potential energy in the form of static pressure. Kinetic energy is added by rotating impellers, while the conversion of velocity pressure to static pressure occurs in downstream stationary components such as diffusers, return channels and volutes.**



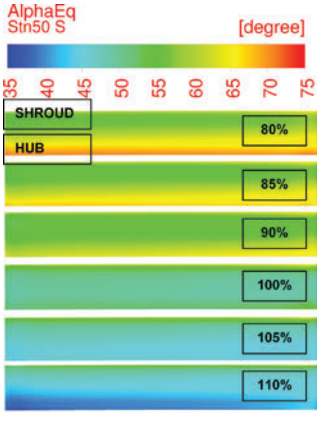
▲ Velocity profile at 90 percent flow for the original stationary component design shows a large low-momentum region at the hub side in the vaneless diffuser and return bend.



▲ Absolute velocity flow angle at the diffuser exit for the original design shows high tangential-flow angles, indicative of low-momentum flow that often leads to formation of stall cells.



▲ Velocity profile at 90 percent flow for the optimized stationary component design shows a much smaller low-momentum region.



▲ Absolute velocity flow angle at the diffuser exit for the optimized design shows greatly reduced high tangential-flow angles.

hub-tapered diffusers. Engineers iterated to a diffuser design that is pinched and tapered on both hub and shroud sides to significantly reduce low-momentum regions that were forming on either side of the diffuser exit at low flow. They reduced the return channel width and redesigned the return channel vanes to match the new flow incidence. CFD results showed that the new design significantly reduced the size of the low-momentum region in the diffuser and return channel. It also considerably delayed the shift of the low-momentum region from the shroud side to the hub side, delaying the onset of stall.

Comparison between the original and optimized designs shows a substantial reduction in the absolute velocity flow angle relative to the radial line at the diffuser exit plane. Highly tangential flow angles greater than 75 degrees generally are indicative of very low momentum, which leads to formation of stall cells in stationary components. The pressure recovery plots for both original and optimized geometries show that the optimized geometry has lower pressure recovery on the overload side but better performance on the surge side. The lower recovery at overload for the optimized geometry is most likely due to the narrow stationary component passages, which results in higher gas velocities and lower pressure recovery. However, this geometry also contributes to improving the flow in the stationary components, resulting in better pressure recovery at lower flow. The

▲ **Industrial centrifugal compressor** COURTESY DRESSER-RAND.

CFD results predicted an improvement in surge margin of approximately 15 percent.

Tests validated CFD simulation prediction of an improvement of about 10 percent in the surge margin of the new design. Flow angle measurements at the diffuser inlet, diffuser exit and return channel inlet confirmed the CFD prediction of a delay in the low-momentum shift. Further, the redesign was successful in maintaining the same head and efficiency levels as the previous design had. About 1.5 percent of overload margin was lost due to the reduced passage areas in the stationary components. However, this was deemed acceptable, as the stage is not expected to be operated at high flow levels close to choke.

Impellers are subjected to inlet and exit flow variations through the stage, and therefore they must be designed to withstand the alternating pressure loads due to these variations in addition to withstanding steady loads. The structural team used ANSYS Mechanical software and in-house tools to ensure that the new design meets the static and dynamic stress requirements.

Dresser-Rand's use of CFD simulation to optimize the stationary components of a new centrifugal compressor design

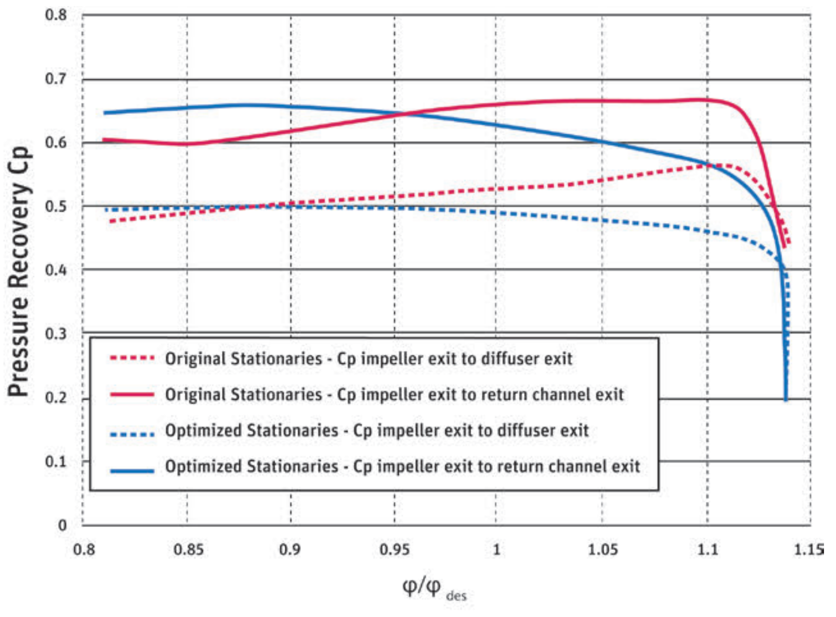
Structural Analysis

Dresser-Rand structural engineers optimize the impeller design to keep the static stresses both below those seen in similar families of impellers and below allowable material yield strengths. The lower the stresses, the faster the impeller can be run. During the design process, engineers also analyze the design to see if there are any possible resonance issues caused by upstream or downstream stationary components.

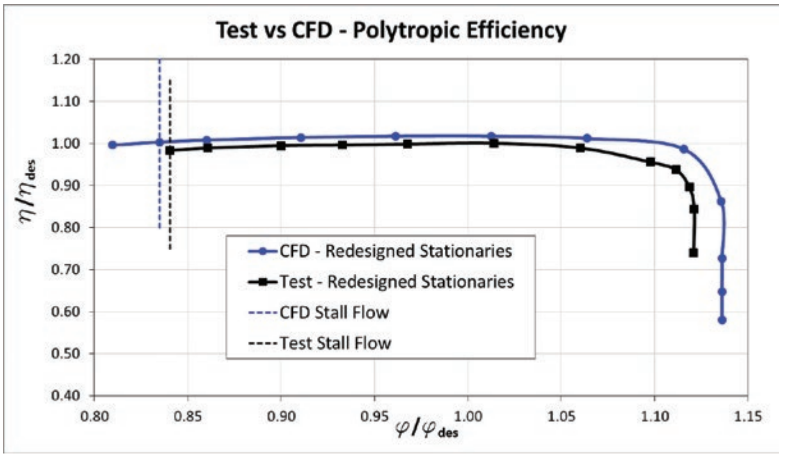
Most of the time, structural damages to the impellers are due to mechanical fatigue. Dresser-Rand follows the in-house dynamic audit process [3] to evaluate the fatigue life of the impellers. The dynamic audit process involves a series of successive analysis runs, starting with a modal analysis and plotting of a SAFE diagram [4] for identifying interferences. This is followed by harmonic response analyses to compute dynamic stress levels in an impeller at identified SAFE interferences. A minimum factor of safety is then computed for all locations in the impeller based on the static and dynamic stresses, material properties and the construction method used for that impeller. The structural team has automated much of the structural design process by writing APDL macros and FORTRAN programs, which have reduced simulation time from more than a week to one to two days per design iteration.

▲ **Typical steady-state plot of impeller**

Tests validated CFD simulation prediction of about 10 percent improvement in the surge margin of the new design.



▲ The optimized design shows better pressure recovery on the surge (left) side of the pressure recovery curve.



▲ CFD results and physical tests provide similar estimates of compressor efficiency.

Performance Parameter	Original Design	Optimized Design
Normalized polytropic efficiency at design flow	1.000	1.001
Normalized polytropic head coefficient at design flow	1.000	1.003
Surge margin	6.1	16.0
Overload margin	13.4	12.1

▲ Testing shows that optimized design improves the operating range.

Reprinted by permission of Dresser-Rand Company

Dresser-Rand delivered a highly efficient compressor with a wide operating range in a small footprint.

accomplished several goals: This new design delayed the transfer of the low-momentum zone from the shroud side of the diffuser to the hub side, and it shows how proper sizing of stationary components in the early stages of the design process can increase the compressor's operating range. The end result is that Dresser-Rand delivered a highly efficient compressor with a wide operating range in a small footprint [2].

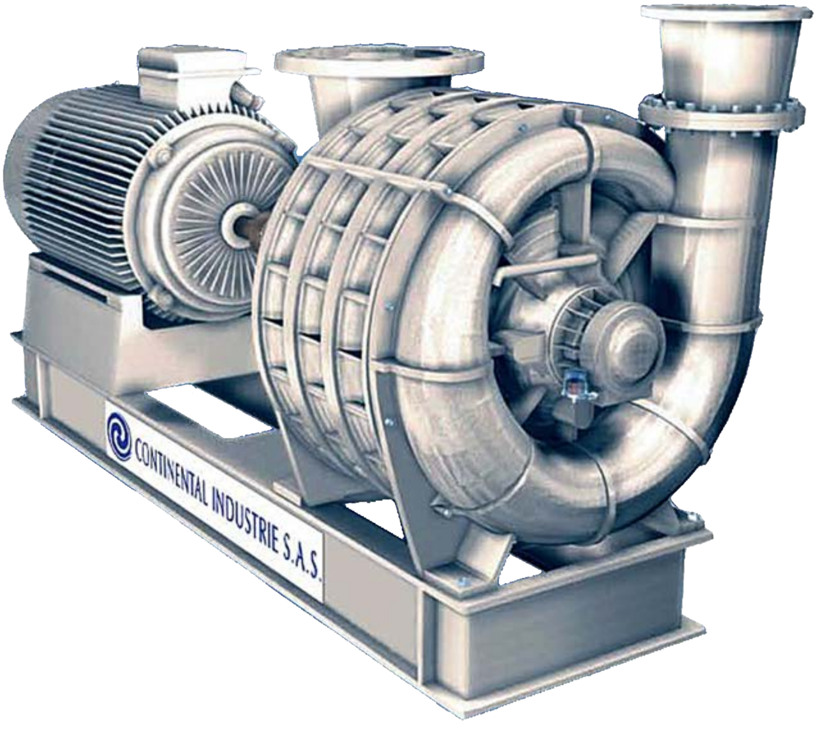
References

[1] Sorokes, J. M.; Pacheco, J. E.; Vezier, C.; Fakhri, S. "An Analytical and Experimental Assessment of a Diffuser Flow Phenomenon as a Precursor to Stall". *Proceedings of ASME Turbo Expo 2012*, **2012**, Volume 8: Turbomachinery, Parts A, B and C.

[2] Fakhri, S.; Sorokes, J. M.; Vezier, C.; Pacheco, J. E. "Stationary Component Optimization and the Resultant Improvement in the Performance Characteristics of a Radial Compressor Stage". *Proceedings of ASME Turbo Expo 2013*, **2013**.

[3] Schiffer, D. M.; Syed, A. An "Impeller Dynamic Risk Assessment Toolkit". *Proceedings of the 35th Turbomachinery Symposium*, **2006**, pp. 49–54.

[4] Singh, M. P.; Vargo, J. J.; Schiffer, D. M.; Dello, J. D. "SAFE Diagram — A Design and Reliability Tool for Turbine Blading". *Proceedings of the Seventeenth Turbomachinery Symposium*, **1988**, pp. 93–101.



BLOWN AWAY

The ANSYS integrated turbomachinery design platform enabled a rotating machinery company to design a centrifugal compressor with a potential for 2 to 5 percent energy savings during wastewater treatment operations. In addition, the company was able to reduce costs and design time in developing a next-generation product.

By **Brice Caussanel** and **Renaud Signoret**, Turbomachinery Engineers, Continental Industrie, Saint-Trivier-sur-Moignans France

Most wastewater treatment plants use naturally occurring microorganisms in wastewater to quickly break down organic matter to form carbon dioxide and water. Aeration plays an integral role in these plants by adding air to the wastewater to promote aerobic biodegradation of the organic pollutants. The compressors that inject this air consume substantial amounts of electrical power to overcome the backpressure of the water height and losses in the air injection system. The amount of power is significant. For example, the approximately 20,000 municipal wastewater treatment plants in the United States consume about 4 percent of all the electrical energy generated in the U.S. [1], and the compression of air for the aeration process is estimated to account for about 60 percent of this power [2].

Aeration is a huge expense for municipal treatment plants, and improving the aeration compressor's efficiency provides an enormous opportunity for cost and energy savings. Continental Industrie has 40 years of experience in research, development and manufacturing of centrifugal blowers and exhaust products. The company's engineers utilized the ANSYS integrated design system for turbomachinery applications to design a next-generation centrifugal compressor for wastewater aeration applications that provides a 2 to 5 percent improvement in efficiency compared to the previous-generation compressor. This should provide savings of 15 kW to 50 kW for the average wastewater plant. Based on operating 2,000 hours per year and a cost of $0.20 per kWh, this would yield an annual savings of $6,000 to $20,000 per year per compressor. Engineers used optimization algorithms

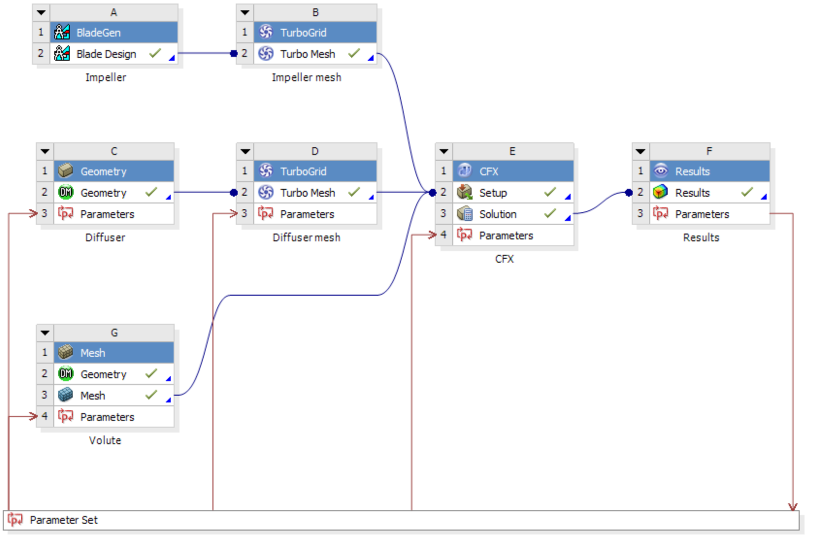
Improving the efficiency of the compressors used for aeration provides an enormous opportunity for cost and energy savings.

to explore 1-D, 2-D and 3-D designs to get the design right the first time while minimizing modeling and computational effort.

PREVIOUS DESIGN METHODS

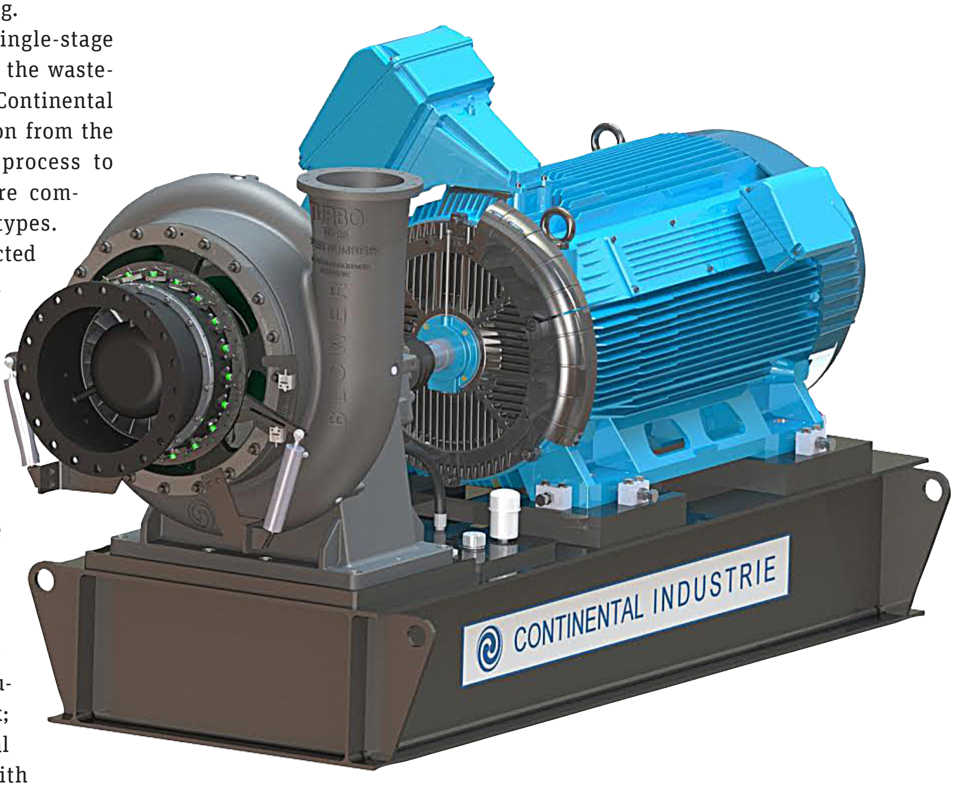
There are many design variables involved in centrifugal compressors, each of which has complex and often interacting effects on the finished product's performance. To design the previous generation of these compressors, experienced designers used empirical methods. The process began with the use of one-dimensional analysis and engineering intuition to obtain an initial design with a reasonable efficiency level. A bench model was then built so that rough performance measurements could be taken. Experienced turbomachinery designers reviewed the test results and made educated guesses on which design changes might be able to deliver significant performance improvements. These designers were able to achieve significant improvements but were not able to fully optimize the design. The full scale prototype did not always meet the design specifications, and this required costly additional cycles of prototype building and physical testing.

To design its newest single-stage centrifugal compressor for the wastewater treatment industry, Continental Industrie utilized simulation from the beginning of the design process to optimize the design before committing to physical prototypes. Continental Industrie selected the ANSYS integrated approach for turbomachinery design for several reasons: The ease of use of ANSYS solutions makes it possible to define a complete workflow and methodology in a short period of time the ANSYS parametric platform allows the team to explore the complete design space to identify the optimal solution with a high level of accuracy, eliminating guesswork; and both flow and structural engineering teams work with

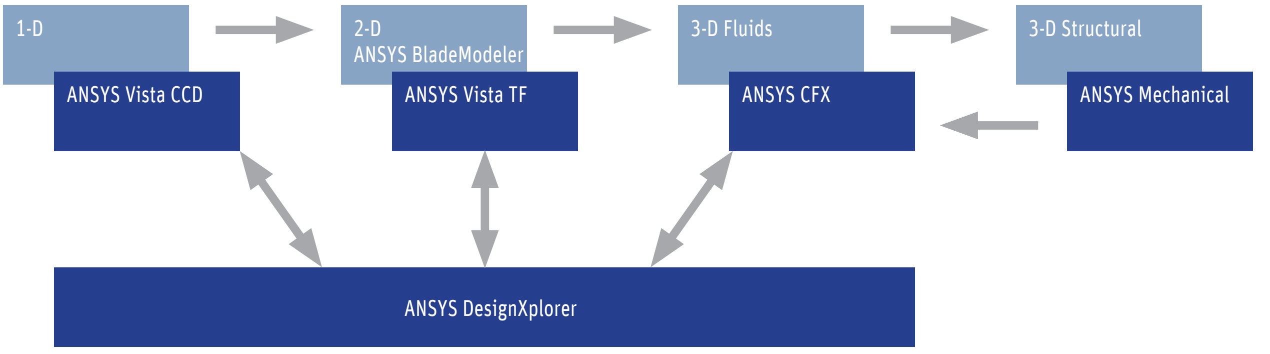


▲ ANSYS Workbench geometry and CFD simulation workflow schematic used in the design of a new compressor

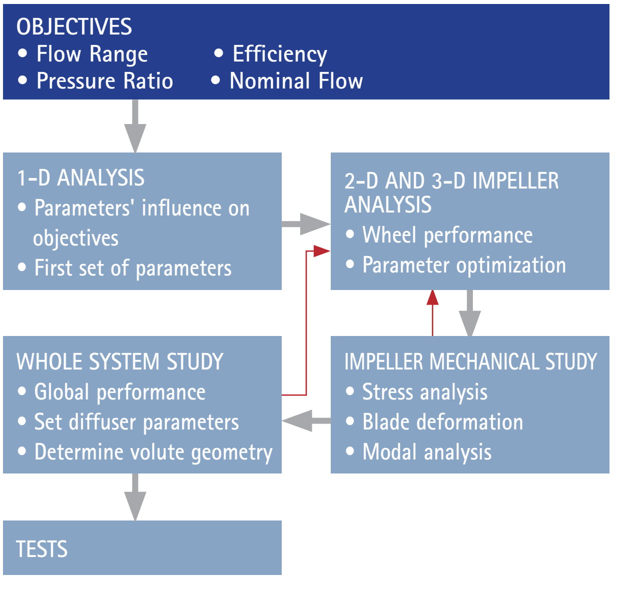
Continental Industrie utilized simulation from the beginning of the design process to optimize the design.



▲ New centrifugal compressor



▲ ANSYS Workbench enabled engineers to easily design and optimize the compressor.



▲ Engineers used simulation to meet design objectives for the new product.

the same design geometry, making it possible to easily incorporate both simulation types into the optimization.

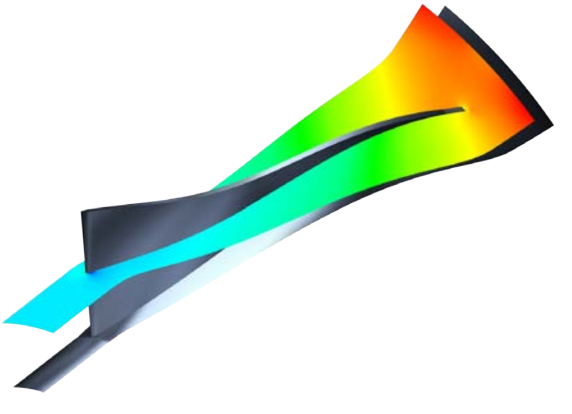
PRELIMINARY DESIGN

Continental Industrie engineers used the ANSYS Vista CCD tool (included with ANSYS BladeModeler software) to perform preliminary design or sizing of the compressor based on input parameters such as the pressure ratio, mass flow rate, rotational velocity and other geometrical constraints. They evaluated about 50 impeller blades manually to gain an informed understanding of the effect of the different parameters, and then used ANSYS DesignXplorer to perform a designed experiment that evaluated about 200 more designs to fully optimize the variations from a 1-D perspective. The very short run times provided by Vista CCD made it possible to evaluate each design in less than a minute.

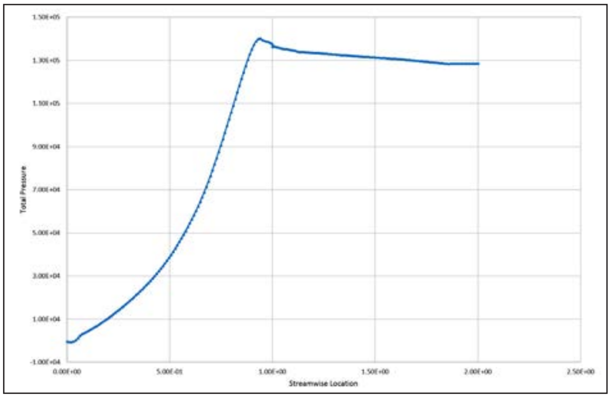
Next, engineers used ANSYS Vista TF to evaluate the 2-D blade row design. The throughflow solution captured many features of a full 3-D flow simulation with much less computational effort. An additional step was performed at this stage to optimize the blade in the context of the impeller. During this stage, Continental Industrie engineers examined 20 different designs and made only very small changes to the parameters, but gained significant improvements in projected efficiency.

COMPLETE 3-D COMPRESSOR DESIGN

The next step involved integrating the impeller into the complete 3-D compressor. Continental Industrie engineers produced the geometry of the complete flow path, including the inlet guide vanes, impeller, diffuser and volute casing in SolidWorks® computer aided design software. After they imported the geometry into ANSYS DesignModeler, the ANSYS Meshing platform generated the mesh in the volute casing fluid volume, and ANSYS TurboGrid automatically produced a hexahedral mesh of all the bladed components — the inlet guide vanes, impeller and diffuser. Using ANSYS CFX computational fluid dynamics (CFD) software to minimize the flow losses through the diffuser and volute, engineers re-optimized the system by employing ANSYS DesignXplorer to perform another experiment, in this case with about 250 designs.



▲ Mid-span pressure field calculated by ANSYS CFX



Total pressure variation through the compressor

STRUCTURAL DESIGN

Engineers also used ANSYS Workbench to link the pressure and temperature predictions from ANSYS CFD to ANSYS Mechanical to evaluate the stress levels and deformation of the impeller wheel and other mechanical components. The structural simulation revealed that the initial impeller design experienced stress values beyond the impeller material yield strength, so engineers increased the thickness of the blade to ensure reliability. Additional CFD calculations were performed

3-D flow simulation using ANSYS CFD allowed Continental Industrie to improve the performance of a centrifugal compressor.

MODELING INTERNAL FLOW-INDUCED MOTION WITH ANSYS FLUID–STRUCTURE INTERACTION — WEBINAR
ansys.com/motion101

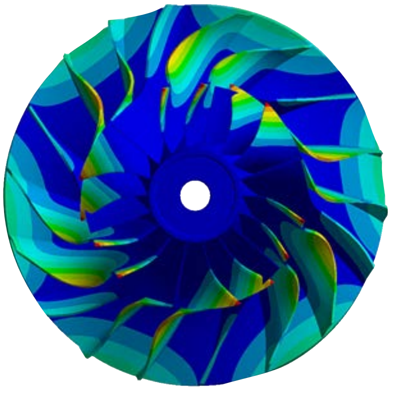
to check the new design at three mass flow rates. The impeller deformation results provided by ANSYS Mechanical were used to avoid contact between impeller blade tips and the shroud. A modal analysis was also performed to investigate the vibrational behavior of the rotating impeller and ensure that it did not have any resonant frequencies that would be excited under normal operating conditions.

By using an integrated design process that optimized the compressor at three separate phases, Continental Industrie engineers were able to deliver 2 to 5 percent higher efficiency than the company's previous generation of wastewater aeration centrifugal compressors. The new compressor can vary flow while maintaining constant pressure, which makes it possible to save even more energy by reducing flow rate to the minimum level required by the process. Continental Industrie also generated substantial cost savings in the process because the entire design was completed by a three-person team, and the first prototype met the company's performance requirements.

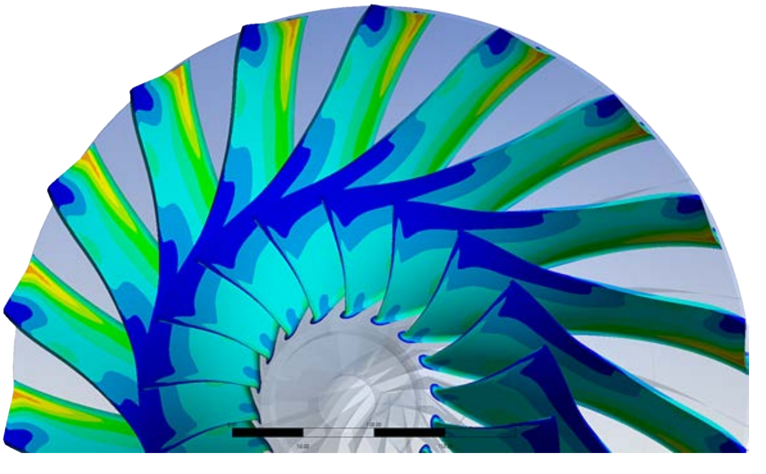
References

[1] Circle of Blue. EPRI Technical Report: Water & Sustainability (Volume 4). U.S. Electricity Consumption for Water Supply & Treatment — The Next Half Century. circleofblue.org/waternews/wp-content/uploads/2010/08/EPRI-Volume-4.pdf (09/22/2015).

[2] Bolles, S. Process Energy Services, LLC. Modeling Wastewater Aeration Systems to Discover Energy Savings Opportunities. processenergy.com/Aeration%20Paper.pdf (09/22/2015).



Modal analysis of the impeller was performed.



Stress field on the compressor blades was simulated to ensure reliability.

1, 2, 3 TURBOCHARGED EFFICIENCY

Specialized advanced simulation tools optimize turbochargers for increased power and fuel efficiency.

By Chris Robinson, Managing Director, PCA Engineers Limited, Nettleham, U.K.

Turbochargers are increasingly used in automotive applications to get more power out of smaller engines. Smaller, augmented engines are more fuel efficient and produce fewer emissions without affecting a driver's perception of handling and performance. The heart of a turbocharger is the compressor. The ideal turbocharger compressor is efficient over a broad operating range and has low inertia while simultaneously complying with package size limitations, robustness and cost constraints.

PCA Engineers designs turbocharger compressors by using multiple levels of highly iterative analyses, from simple 1-D and 2-D analyses to highly sophisticated, transient, multiphysics 3-D simulations. The locus of the compressor operating point within the engine can be mapped onto compressor characteristics that plot compressor efficiency and pressure ratio against rotational speed and throughput of air.

PRELIMINARY ANALYSIS CONFIRMS DESIGN TARGETS ARE MET

PCA starts a turbocharger compressor optimization with a set of design points from the key operating zones depicted in the compressor map. The customer provides the design points along with other constraints, such as material, manufacturability and package size. PCA's designers then evaluate the performance of proposed design alternatives against these targets. This work is done with ANSYS Vista CCD, a preliminary design tool embedded within the ANSYS Workbench environment. Given the aerodynamic duty — such as pressure ratio, mass flow, and rotational speed, and fixed geometric constraints such as inducer hub diameter and vane thickness — Vista CCD calculates a suitable 1-D compressor geometry and the associated performance map. Engineers then superimpose the map from Vista CCD onto the client-provided targets to identify any potential shortcomings in the prototype. The differences between the 1-D compressor map and the targets are then used to guide revisions to design parameters. This process is repeated iteratively until a satisfactory 1-D design has been achieved. From the 1-D compressor geometry, PCA's designers will have a good idea of how a design fits within their range of experience and what problems are likely to be encountered in improving it.

In the next phase of the design, engineers further refine the geometry of the blade by launching ANSYS BladeModeler from Vista CCD, defining the basic geometry of the blade in that code. Next, the model is exported to the ANSYS Vista TF 2-D throughflow solver. The throughflow step solves the circumferentially averaged inviscid equations of motion to produce a solution for the proposed design that can be assessed against performance criteria. ANSYS Vista TF incorporates established empirical models for losses and deviation; it can, within a few seconds, capture many of the important features of a full 3-D flow simulation. All of the work is done within Workbench, facilitating rapid improvements in blade design in a process that is amenable to automatic iteration and optimization.

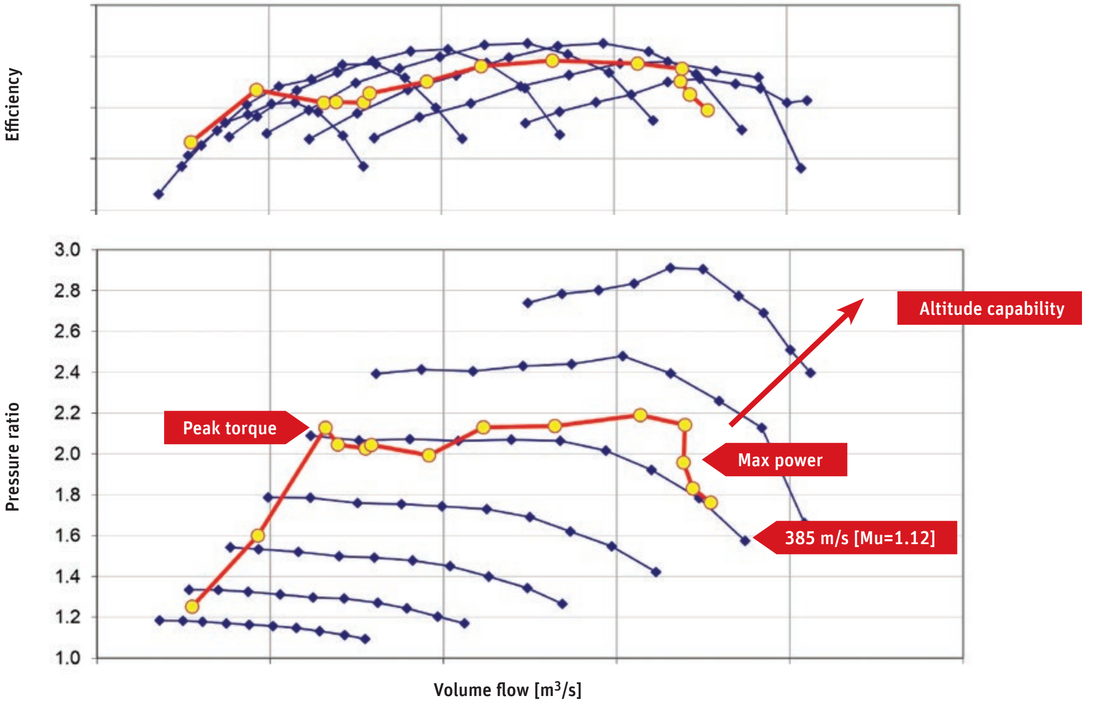
FLUID DYNAMICS FOR EFFICIENCY

Once the preliminary blade design is complete, PCA's engineers use ANSYS TurboGrid to produce a hexahedral mesh for the impeller and diffuser prior to full 3-D computational fluid dynamics (CFD) analysis. After stage analysis is complete, they design the volute, the component that receives high pressure air from the compressor via the diffuser. PCA greatly reduced the time required to define the volute's geometry by automating design iterations within ANSYS Workbench, producing a CFD-ready model of the gas-swept surfaces in a single operation.

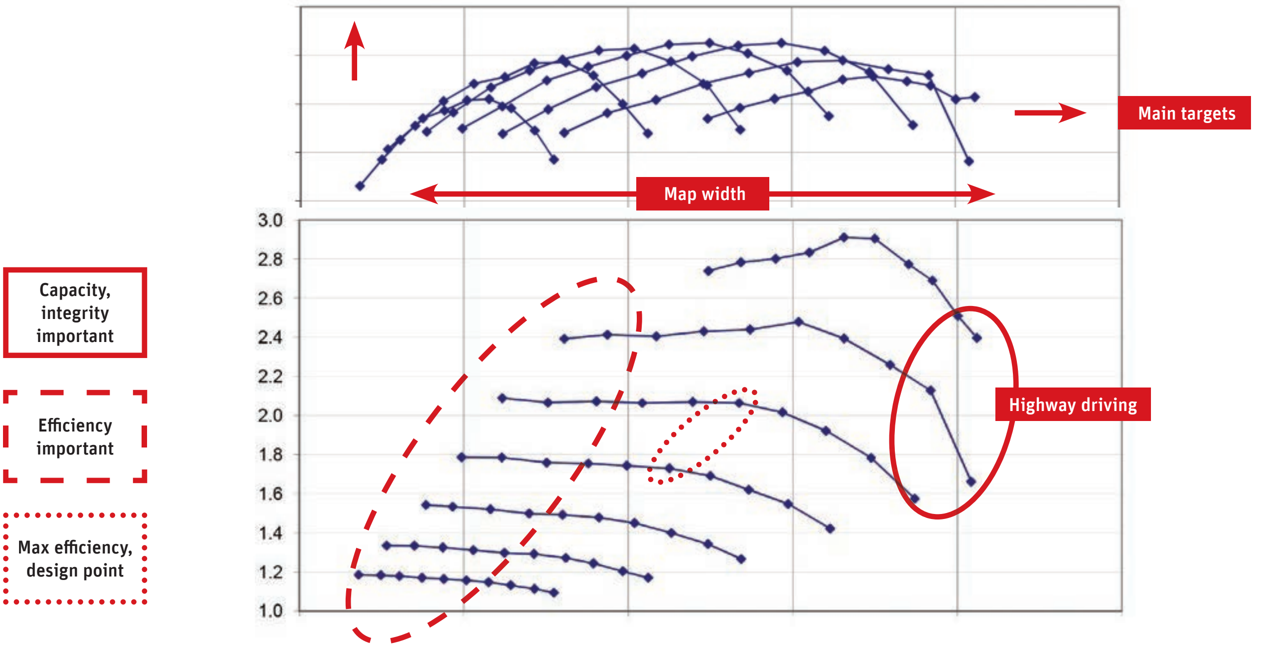
PCA's engineers then assemble the virtual design and generate the compressor map data by running a full CFD simulation at several flow rates for each speed of interest to verify that the new stage meets the customer requirements. By taking into account the compressor's full 3-D geometry environment for the final analysis, PCA can evaluate aspects of the design that cannot be captured in the 1-D or 2-D analysis, such as the effects of tip clearance. Tip clearance tends not to scale linearly, as do other dimensions, so small compressors have to be run at relatively high clearance levels, which limits both efficiency and range.

LONG LIFE AND ROBUSTNESS

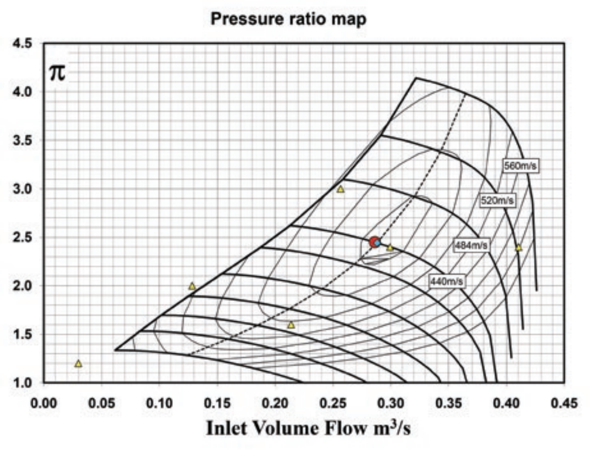
A compressor's mechanical performance is rated on whether or not it delivers the



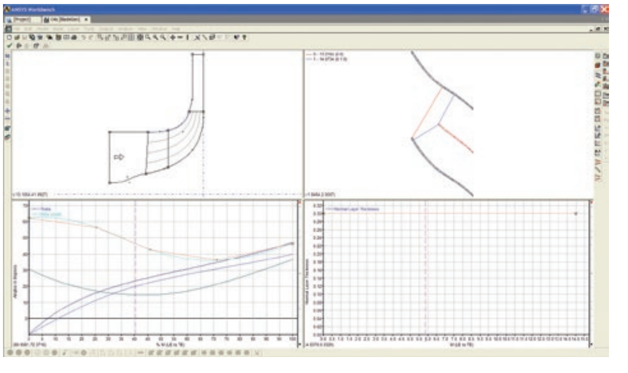
▲ A gasoline engine turbocharger compressor performance map, as measured on a test rig, is used to assess compressor suitability for a given application by showing pressure and efficiency vs. volumetric flow rate. Red lines show approach to choke, or maximum capacity, in which volume flow becomes constant and both pressure ratio and efficiency drop off.



▲ Compressor performance maps illustrate operating regions with the greatest impact on compressor design. The dashed line zone represents idle to cruise typical of city driving, conditions for which impeller inertia (turbo lag) is a concern. The solid line zone represents highway driving for which impeller stresses are maximized: This is the focus of most of the design effort.



▲ Typical turbocharger compressor performance targets provided to guide a compressor design

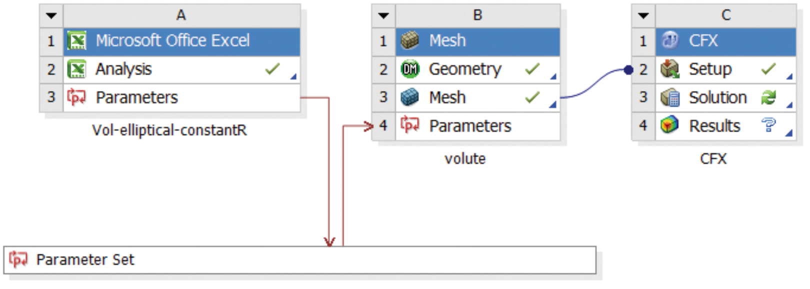


▲ Typical output from BladeGen blade geometry designer module in ANSYS BladeModeler

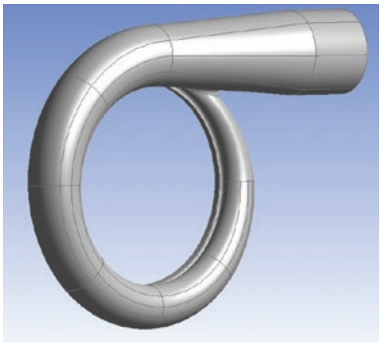
expected service life and integrity: Turbochargers are expected to last for the life of a vehicle. Blade stress is proportional to the square of tip speed, so addressing mechanical issues at conditions that require the compressor to run faster (such as high altitudes) are essential in developing designs that have adequate life and robustness. To avoid failure by high cycle fatigue is essential, and the lowest natural frequency is usually first flap, which is the mode that would be excited if you "ping" the tip of the blade. This first vibrational mode must be at a high-enough frequency that no vibration is triggered by harmonics from unbalanced forces from the rotating shaft or flow features upstream/downstream of the impeller.

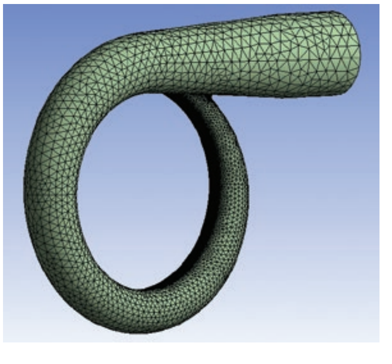
PCA performs structural analyses to test for vibrational modes in the same ANSYS Workbench environment, eliminating the need to transfer and recreate the geometry, thus reducing the opportunity for error. The aerodynamic pressures calculated by CFD can be applied as loads for structural analysis to augment the centrifugal loads that are almost always dominant for small turbochargers. In addition to being concerned about stress levels and blade natural frequencies of vibration, designers must focus on the deflection of the rotor, both radially and forward, due to the asymmetric shape of the disc. They usually mitigate this by using backface extension as a counterweight, reducing deflection and bore stress.

Using ANSYS Workbench tools, PCA Engineers can optimize the design of a turbocharger compressor long before committing to expensive prototype hardware and tests. A key advantage is that the complete design process, including 1-D analysis,



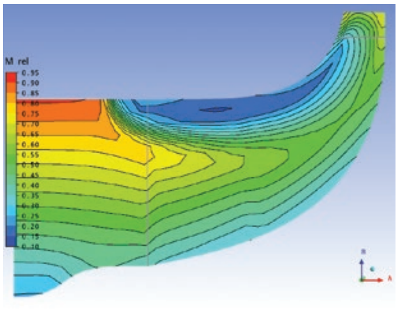
Outline: No data			
	A	B	C
1	ID	Parameter Name	Value
2	⊟ Input Parameters		
3	⊟ volute (B1)		
4	P39	a.90	4.65
5	P41	a.180	6.5761
6	P37	a.0	1.0395
7	P43	a.270	8.054
8	P44	a.315	8.6994
9	P40	a.135	5.6951
10	P42	a.225	7.3523
11	P38	a.45	3.288
12	P45	a.360	9.3
13	P46	b1.0	1.0395
14	P47	b1.45	3.288
15	P48	b1.90	4.65
16	P49	b1.135	5.6951
17	P50	b1.180	6.5761
18	P51	b1.225	7.3523



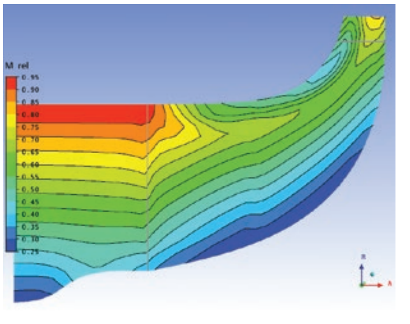


▲ An Excel® macro drives the iterative geometry definition in ANSYS Workbench. The design can be optimized in response to design parameters chosen by the designer.

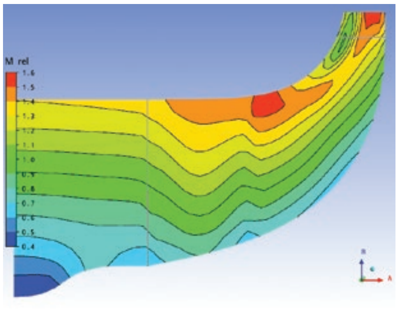
385 m/s (near surge)



440 m/s (peak efficiency)



495 m/s (near choke)

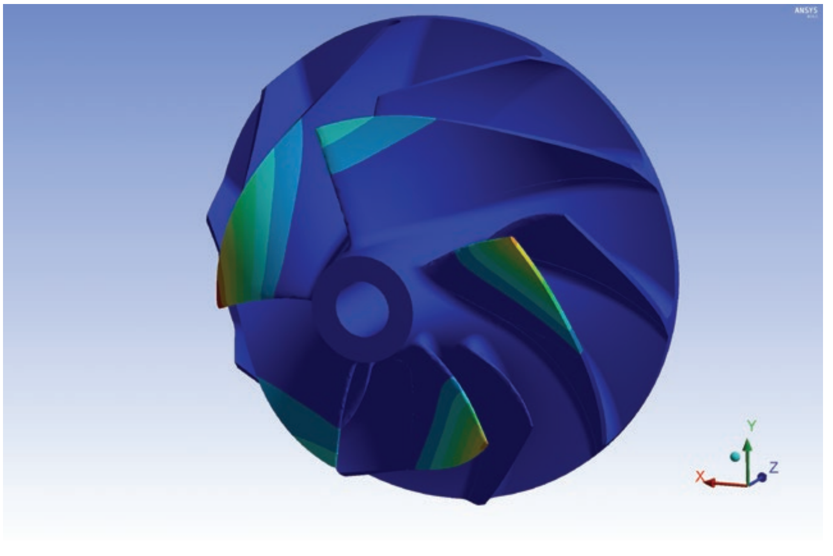


▲ Flow conditions at three different operating points between surge (top) and choke (bottom). This demonstrates the performance impact of tip-clearance effects on velocity at various flow rates. Speeds shown refer to tip speed.

The complete design process is performed within a single environment. This streamlined approach reduces both engineering effort and time to market.

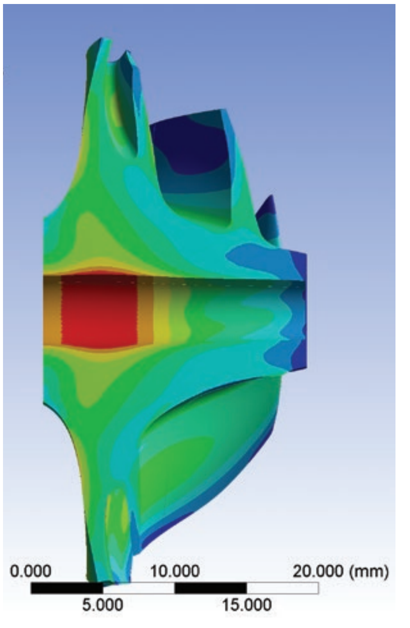
2-D analysis, 3-D geometry definition and meshing, 3-D flow simulation, and structural analysis, are all performed within a single environment. This streamlined approach reduces both engineering effort and time to market by eliminating the need to move or duplicate geometry, meshing and physical parameters information from one environment to another.

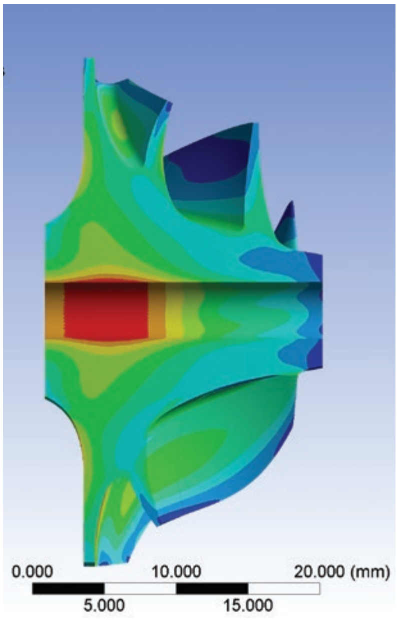
PCA plans to further accelerate the design process by automating more of the simulation through macros and scripts and by transferring its successful, automated optimization experience into the Workbench environment.



▲ ANSYS Mechanical prediction of blade deflection at first vibrational mode. This mode is engineered so that its frequency is high enough to prevent vibration from being triggered during compressor operation (first flap frequency).

Addressing mechanical issues at conditions that require the compressor to run faster are essential in developing turbocharger designs that have adequate life and robustness.





▲ Maximum stress is in the bore and is arranged to be away from the contact zone. As the impeller runs up in speed, it leans forward at the rim and shortens axially.

PHOTO COURTESY ANDRITZ HYDRO

FORCE OF NATURE

As a leader in the global hydropower industry, ANDRITZ HYDRO has built a reputation for product quality, design robustness and innovation. Head of R&D Engineering Methods Mirjam Sick discusses how engineering simulation has supported the company's groundbreaking design efforts for the past 25 years — and looks toward the future of simulation in her industry.

By *ANSYS Advantage* Staff

Headquartered in Vienna, Austria, ANDRITZ HYDRO is a global supplier of electromechanical systems and services for hydropower plants. As one of the leaders in the world market for hydraulic power generation, the company has installed more than 30,000 turbines globally, which are capable of producing 400,000 megawatts of output.

To keep pace with changes in international energy needs, growing environmental concerns and tightening government regulation, ANDRITZ HYDRO relies on engineering simulation to arrive at innovative new turbomachinery designs that deliver reliable, robust performance for customers around the world. This longtime ANSYS customer leverages the power of simulation to help customers retrofit aging hydropower plants with new technologies as well as to troubleshoot performance issues for existing power generation systems.

As head of R&D Engineering Methods, Mirjam Sick manages the organization's engineering simulation efforts and other critical design activities. Recently, *ANSYS Advantage* asked Sick about the competitive challenges that ANDRITZ HYDRO faces today, how her team uses engineering simulation to meet these challenges, and her perspective on the future of hydropower engineering.

Our engineers must analyze performance over time, under conditions that are constantly changing.

What are the biggest challenges facing the hydropower industry today — and what role do engineers play in overcoming them?

The global hydropower industry has changed dramatically in the past decade, reflecting changes in the broader energy industry. As other renewable energy sources, such as solar and wind, have grown in capabilities, the energy grid looks very different today. This has resulted in enormous demand swings for hydropower plants. At one time, demand was stable and plants were running continuously — but today there is a great deal of variability. This means that hydropower turbines, pumps and hydraulics must be designed for flexible operation and performance that is far more dynamic in nature. Our engineering teams must now analyze how equipment performs under off-design conditions that we would not have considered 10 years ago — including dynamic loads and flows that are close to stability limits. We must build our machinery to change quickly and efficiently to new operating modes. It's not enough to design for the best-possible condition; our engineers must analyze performance over time, under conditions that are constantly changing. We need to consider how varying loads and flows affect lifetime performance. We need to design turbines that are incredibly robust and reliable. ANDRITZ HYDRO has an outstanding reputation for product quality, and we are working hard today to uphold that tradition, even as our engineering challenges have grown in complexity.

How has your use of engineering simulation evolved to help you meet these new performance demands?

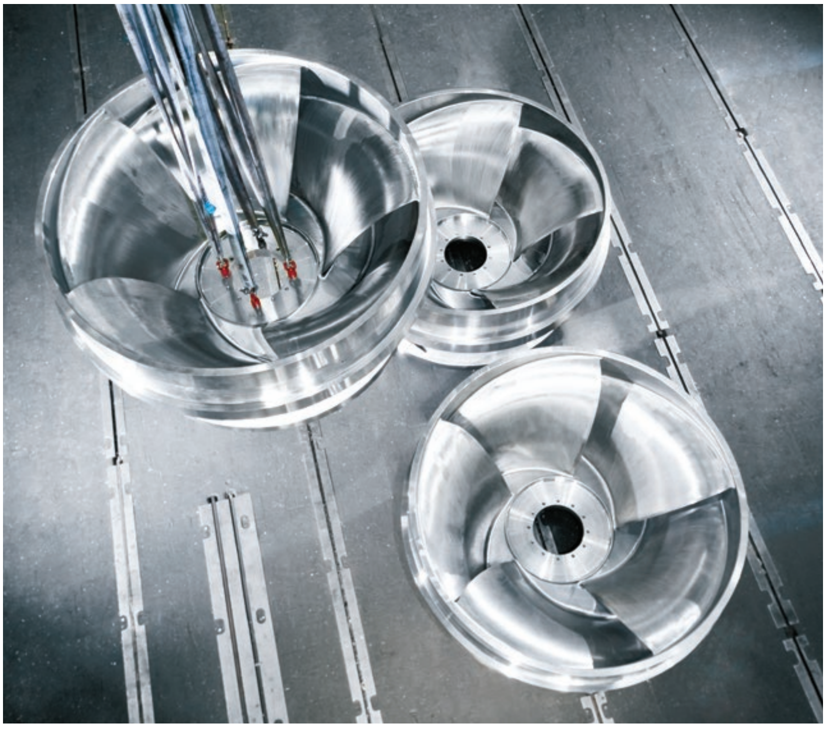
Our engineers are doing more time-dependent simulations instead of focusing only on steady-state equipment performance. We're looking at the interactions of static and rotating parts, instead of focusing only on the parts that are moving. We're performing dynamic stress analysis that allows us to increase our understanding of the impacts of changing flows and loads. Time-dependent methods are required to analyze starts and stops of generators and turbines. In addition, our engineering team is conducting one-way coupling for computational fluid dynamics (CFD) and finite element method (FEM) studies. Electromagnetics (EM) simulation tools are helping us to optimize the performance of generators over their lifetimes.

Fortunately, as our engineering needs have become more advanced, engineering simulation tools have grown in their sophistication and their ability to support these kinds of in-depth, multiphysics analyses. We've been very pleased with the way ANSYS software has improved over time to add these critical capabilities. Designing our advanced turbines, pumps and hydraulics without simulation tools would not be possible. Fortunately, ANSYS not only provides these simulation tools, but delivers excellent support for our analyses. The software enables us to replicate an incredible level of product complexity, as well as very challenging operating conditions, in a low-risk virtual design environment.

How does ANSYS software help create a competitive advantage for ANDRITZ HYDRO?

Our company is known in the industry for our engineering knowledge and our expertise in R&D. Worldwide customers come to us for superior new designs as well as for ideas to improve the

Software from ANSYS enables us to replicate an incredible level of product complexity.



▲ Pump impellers used for irrigation of agricultural land COURTESY ANDRITZ HYDRO.

performance of their existing hydropower systems. We rely on ANSYS software to *show* customers that expertise in a visual way. If we are developing a new turbine system or refurbishing an older plant, we often show customers ANSYS simulation graphics that demonstrate exactly how the new machinery will work — or exactly where current performance problems arise, in the case of a plant retrofit. Our ANSYS tools help us interact with customers in a very professional and serious way, demonstrating our engineering knowledge in a straightforward manner. The fact that we have validated all our designs via ANSYS gives our customers great confidence in ANDRITZ HYDRO. They trust that our systems will work as promised.

ANDRITZ HYDRO is a good example of a global business, with a global engineering team. How do you facilitate collaboration among your engineers?

In my view, it's critical to have all our engineers using the same set of tools, including ANSYS software. This allows us to standardize our processes and our problem-solving approach across the entire organization — no matter what specific customer challenge our individual engineers are focused on. We have been using ANSYS tools for about 25 years, and today we have more engineers doing simulation than ever before. That level of product innovation creates obvious strategic benefits, but it also means that we must continually share knowledge and encourage collaboration. Our global engineering team comes together in phone calls and meetings on a regular basis. We find it even more important that, every day, team members are applying the same toolkit — and advancing our collective knowledge of how to best leverage engineering simulation as a discipline.

We need to model complex problems much faster to arrive at new, flexible hydropower systems.

What trends do you see in hydropower engineering over the next five to 10 years?

While other renewable power generation systems are receiving a lot of attention, hydropower is still the most cost-efficient and readily available source of renewable energy. Today, only about 20 to 25 percent of global hydropower resources have been developed, which means there is huge growth potential in our industry.

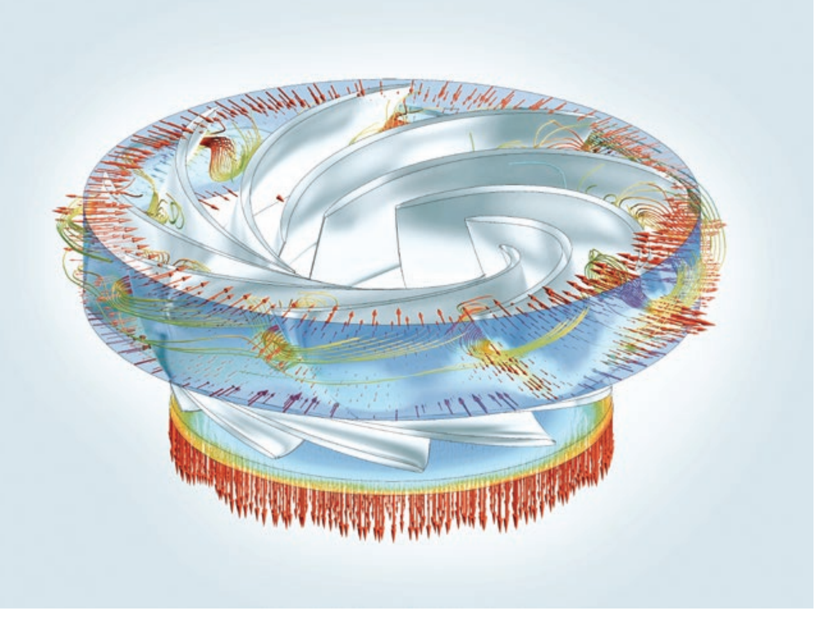
One important opportunity is to develop tidal turbines, or underwater windmills, that rotate based on the natural movement of tides. While this is a promising area, it represents an enormous engineering challenge. It is difficult to accurately assess the changing environmental loads of this type of power generation system. Any system would have to be designed for outstanding strength, since it is subject to rough conditions and, at the same time, difficult to access and maintain. I think over the next decade we are going to see hydropower engineers master this challenge and develop tide-based hydropower plants that help the world address its energy challenges. This will be exciting to witness.

In more traditional hydropower installations, I think the biggest engineering challenge will be designing systems that can go from zero power to 100 percent power very quickly, without any loss in long-term reliability. This means minimizing cavitation, vibration, and other sources of stress and wear within our machines. As consumer needs continue to fluctuate, and the energy grid evolves in its composition, the pressure to operate at variable speeds will only increase.

Engineering simulation software must expand its capabilities for us to meet these challenges. Time-dependent studies, turbulence modeling, flow instability and separation, vortex modeling, heat transfer in generators — we need to model these types of complex problems much faster to arrive at new, flexible hydropower systems. Simulation software from ANSYS has made tremendous strides since ANDRITZ HYDRO started using these tools 25 years ago, and I'm confident that ANSYS software will continue to evolve to meet our increasingly sophisticated user demands.

How would you describe your relationship with ANSYS?
ANDRITZ HYDRO has enjoyed a very close relationship with ANSYS. In fact, I would call it a "development partnership." We have obviously benefited from our ability to apply ANSYS tools to validate our engineering designs. In turn, we have helped ANSYS understand the needs of very advanced users, and this has helped them to improve the modeling capabilities of their tools. For new and complex applications, we have been collaborating by providing test cases, testing alpha versions of models, and working with dedicated research licenses. Without ANSYS, we would be building suboptimal machines. By using ANSYS, and partnering with them to provide ongoing feedback, we can offer our customers a very high level of design robustness and product confidence. I would say we have a very good partnership with ANSYS that has benefited ANSYS, ANDRITZ HYDRO and our global customers.

Without ANSYS, we would be building suboptimal machines.



▲ **Simulation of a Francis turbine** COURTESY ANDRITZ HYDRO.

Optimizing System Performance

While ANDRITZ HYDRO is known for its innovative hydropower products, the company also applies its engineering knowledge to help troubleshoot problems in systems manufactured by competitors. Recently, the engineering team at ANDRITZ HYDRO was called upon to solve performance problems at a hydropower plant located in New Zealand.

"Because we use ANSYS software to simulate systems-level performance on a standard basis, we were able to quickly model this existing system. We observed some damage and suspected that a mistake in the basic design was causing wear issues. We identified the root cause of the problem — which was cavitation in the system's guidevanes," said Mirjam Sick of ANDRITZ HYDRO. "We changed the design to minimize the cavitation, and the problem was solved in a very rapid, straightforward manner. I love the flexible way we are able to use ANSYS tools today to conduct root-cause analysis and add value for our customers."

I love the flexible way we are able to use ANSYS tools today to conduct root-cause analysis and add value for our customers.

FLUID–MECHANICAL SYSTEMS SIMULATION

SHAKING ALL OVER

Multiphysics simulation solves a vibration issue in a Francis turbine.

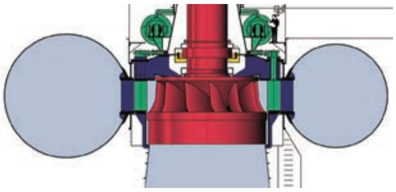
By Björn Hübner, Development Engineer, Voith Hydro Holding GmbH & Co. KG, Heidenheim, Germany

Courtesy Voith.

Strong vibration and pressure pulsation in hydraulic turbomachinery may be quite harmful to machine performance, longevity and safety. It can cause noise, cracks or even machine failure.

Voith Hydro — one of the world's leading suppliers of hydroelectric equipment, technology and services — observed strong vibrations that had the potential to cause fatigue cracking in the guide vanes of a Francis-type water turbine. In a vertical-shaft Francis turbine, water enters horizontally into a spiral-shaped pipe (spiral casing), which wraps around the circumference of a rotating runner. Stationary guide vanes regulate and direct the water to the periphery of the runner. Inside the runner channels, the

Voith Hydro observed strong vibrations that can cause fatigue cracking in the guide vanes of a Francis water turbine.

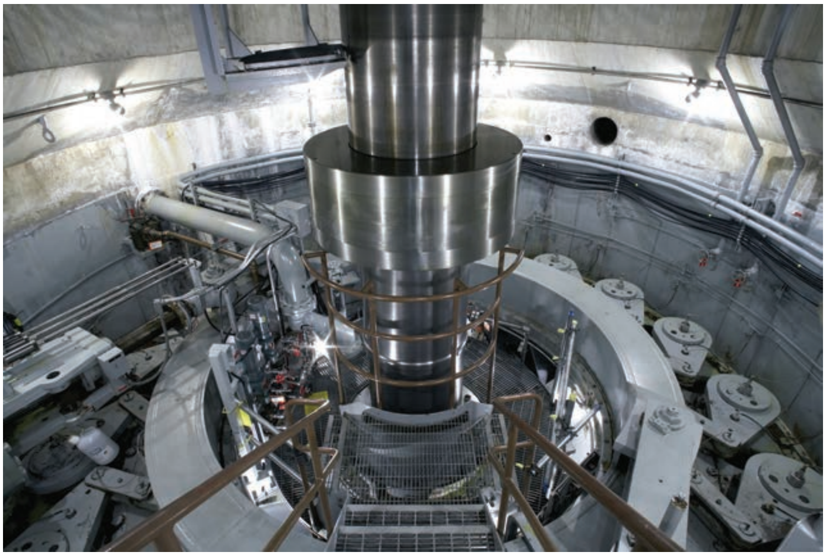


Runner and shaft (red), guide vanes with servo motor (green). *Courtesy Voith*.

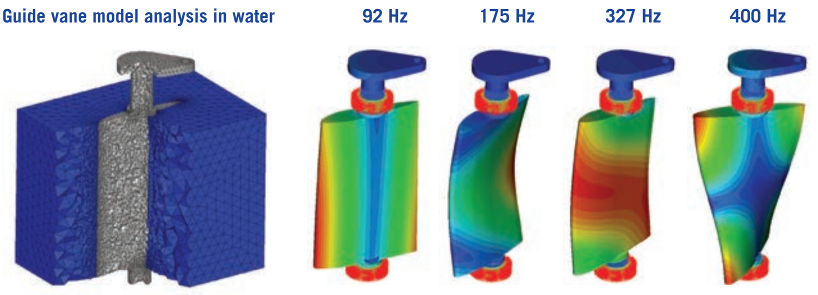
potential energy of the water pressure is transformed into torque, which causes the runner and attached shaft and generator to rotate. Water exits the runner vertically downward into the draft tube where remaining kinetic energy is transformed into additional pressure head.

Using structural simulation, the Voith engineering team ruled out self-excitation and resonance of the guide vanes as the cause of vibration. Employing computational fluid dynamics (CFD), they determined that there was vortex shedding on the runner blades, but not on the guide vanes, that could cause the vibration. This particular machine consists of 24 guide vanes and 13 runner blades; it has an operating speed of 75 rpm. Vibration measurements revealed that all guide vanes vibrated at exactly the same frequencies within the range of 290 Hz to 305 Hz, but it was not possible to perform vibration measurements on the runner blades during operation.

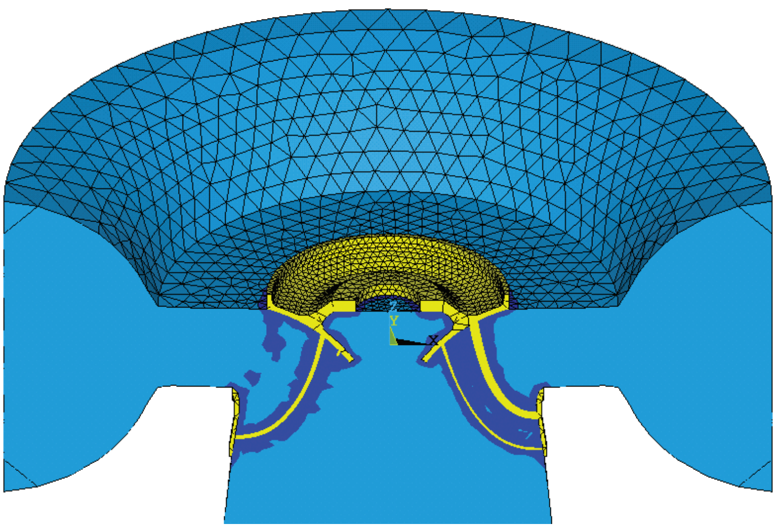
To establish how vortex shedding on the runner was affecting the guide vanes, the team used acoustic fluid–structure interaction with a finite element model of the runner in a water domain. The model used fluid finite elements to couple the dynamic behavior of the runner and water passage. The results proved that excitations at the runner blades' trailing edge were causing the vibration. The simulation matched the measured vibration frequency of approximately 300 Hz. After changing the trailing-edge shape of the prototype runner blades to minimize vortex shedding, observed vibrations were substantially reduced.



Physical measurements of guide vane vibration. *Courtesy Voith*.



Modal analysis showed that guide vane natural frequencies are far from measured frequencies of vibration. *Courtesy Voith*.



Vibro-acoustic finite element model of runner in a simplified water domain. *Courtesy Voith*.

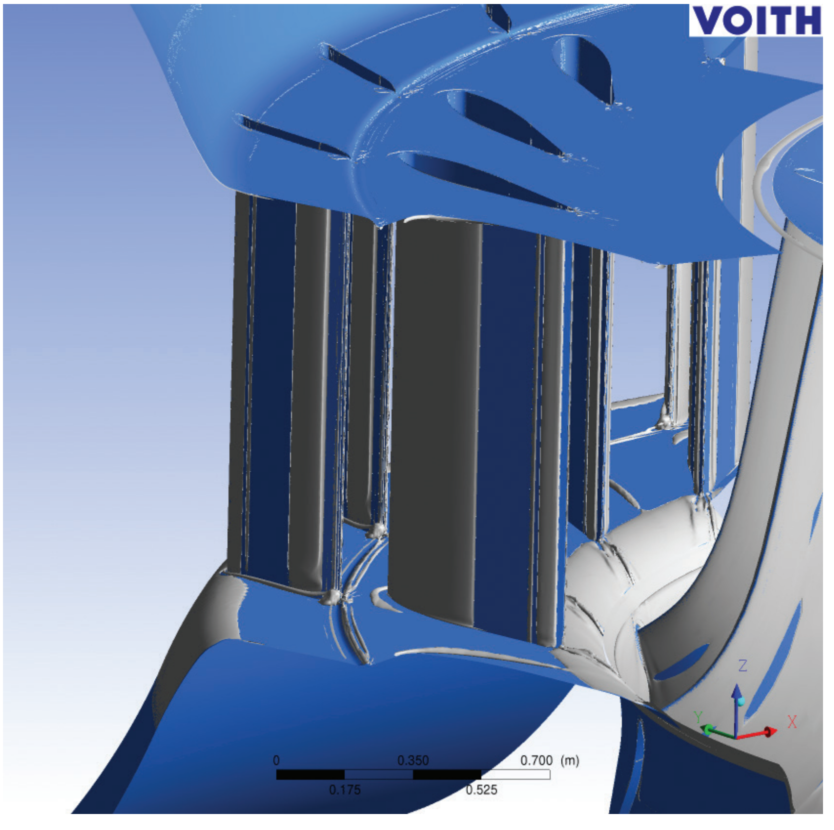
SELF-EXCITED VIBRATIONS AND RESONANCE

To determine the cause of vibration, Voith engineers began by examining the possibility of resonance effects or self-excited vibrations of a guide vane that would occur at a natural frequency. They used ANSYS Mechanical to create a finite element model of the guide vane in water and calculated the first four mode shapes

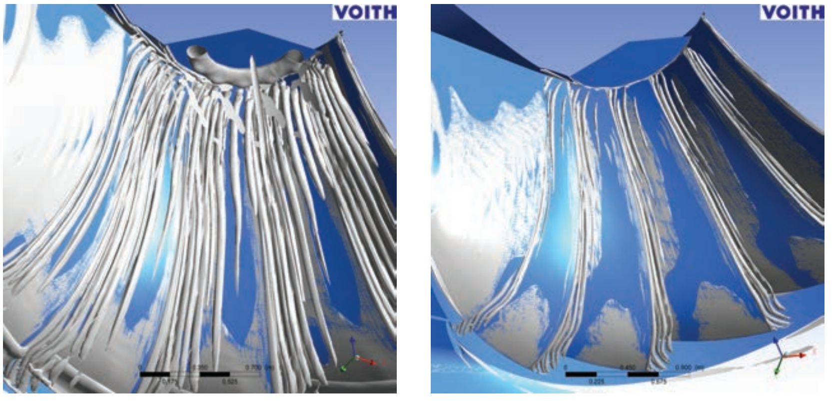
HARMONIC VIBRATIONS AND ACOUSTICS

using undamped modal analysis. Engineers found that there were no natural frequencies close to the observed vibrational frequencies, indicating that guide vane resonance or self-excitation was not present. This finding was confirmed by physical measurements that showed all guide vanes vibrated within the same narrow frequency range, even though small differences in geometry and bearing conditions caused each of the guide vanes to have somewhat different natural frequencies.

To determine how vortex shedding on the runner affected the guide vanes, the team used acoustic fluid–structure interaction.



▲ Vortex shedding was not seen around guide vanes by looking at the Q-criterion that visualizes iso-surfaces of the second invariant of the strain-rate tensor, enclosing spatial regions with minimum pressure to identify vortices in a flow field. *Courtesy Voith.*



▲ CFD simulation of runner blades showed vortex shedding, visualized by two different Q-criteria. *Courtesy Voith.*

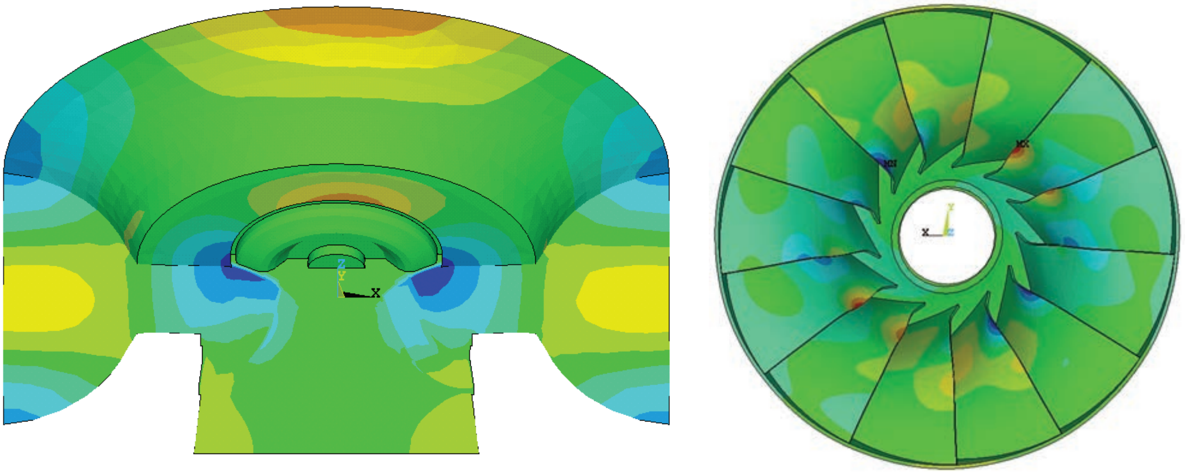
VORTEX SHEDDING

Voith performed unsteady CFD analyses with ANSYS CFX to investigate the possibility of vortex shedding at the guide vanes. The trailing edge used on the guide vanes was designed to prevent vortex shedding, and the analysis showed no sign of shedding. Therefore, the engineers concluded that the problem was not caused by vortex shedding at the guide vanes.

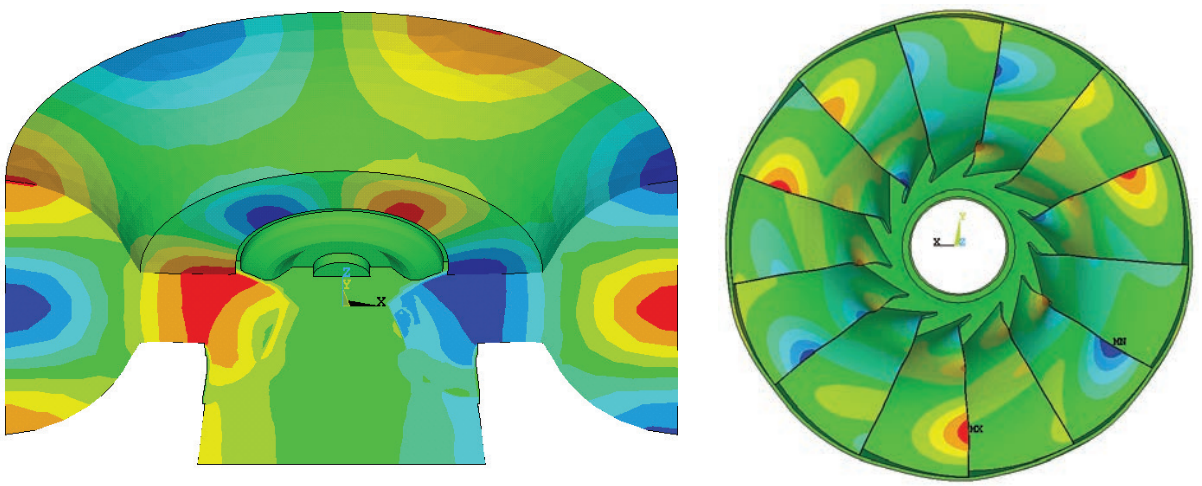
Next, the team performed unsteady CFD analyses at the runner blades. Because the manufactured trailing-edge shape may deviate slightly from the as-designed shape, engineers analyzed both the as-designed chamfered edge as well as a blunt trailing edge. Vortex shedding was clearly observed around 220 Hz for the blunt edge and 370 Hz for the chamfered trailing edge. For a rigid runner, vortex shedding frequencies at different blades and along the trailing edge of a single blade typically differ despite the fact that all of the guide vanes vibrate at the same frequency. The reason is that if some natural frequencies of the mounted runner in water are located in the frequency range of the vortex shedding, and if the corresponding mode shapes include trailing-edge bending, then the vortex shedding frequency may lock in and resonate at this natural frequency. The lock-in effect can cause large amplitude vibrations.

COUPLED DYNAMIC BEHAVIOR

However, vortices that separate from the runner blades move downstream into the draft tube and do not affect guide vanes directly. Thus, even with amplified vortex shedding due to lock-in effects, there must be an additional explanation for the propagation of the pressure pulse in the upstream direction to the guide vanes. Both modal and harmonic response analyses were performed with ANSYS Mechanical to investigate the coupled dynamic behavior of the entire runner and water passage using a vibro-acoustic model of the runner in a simplified water domain created using fluid elements. The finite element model included a rotating frame of reference of the runner and a simplified model of the stationary parts with full rotational symmetry. The runner structure was fixed in the axial and circumferential direction at the connection to the shaft,



▲ Pressure field (left) and axial runner displacement (right) of vibro-acoustic mode shape with two diametrical node lines at a natural frequency of 301 Hz. *Courtesy Voith.*



▲ Pressure field (left) and axial runner displacement (right) of vibro-acoustic mode shape with three diametrical node lines at a natural frequency of 325 Hz. *Courtesy Voith.*

ANALYZING VIBRATION WITH ACOUSTIC–STRUCTURAL COUPLING

and a fluid–structure interface was coupled to the runner structure and acoustic fluid domain. This simplified modal analysis of the undamped vibro-acoustic model provided mode shapes and corresponding natural frequencies. Multiple natural frequencies were detected close to the measured frequency range of the guide vane vibrations. Most of the associated vibro-acoustic mode shapes exhibited large bending displacements at runner-blade trailing edges as well as strong pressure fluctuations in the guide vane area.

Harmonic response analysis was performed to get a clearer picture of the vibro-acoustic effects in the area of the runner and distributor. The runner was excited by rotating force patterns with distinct numbers of diametrical node lines. Each natural frequency has a particular mode shape defined by the number of diametrical node lines. At each runner blade, a single force acts on the trailing edge perpendicular to the blade surface. The results revealed vibro-acoustic resonances with large bending displacements and high pressure pulsations. Both pressure and displacement criteria exhibited clear resonance peaks at 295 MHz for a mode shape with three diametrical node lines and 306 Hz for a mode shape with seven diametrical node lines, which is close to the measured vibration.

The results of the harmonic response analysis together with modal analysis indicate that lock-in effects based on coupled vibro-acoustic resonance conditions synchronize and amplify vortex shedding. The corresponding vibro-acoustic mode shapes propagate and amplify pressure pulsations within the rotating and stationary components of the turbine. The pressure pulsations cause synchronized guide vane vibrations at the natural frequencies of vibro-acoustic mode shapes. The problem was solved by a modified trailing-edge shape that minimized and de-tuned vortex shedding at the runner blades, substantially reducing the guide vane vibrations.

Determining and solving this vibration issue may not have been possible using a single physics. It required understanding all physics involved and applying them appropriately to the problem at hand.

Reference

Hübner, B.; Seidel, U.; D'Agostini Neto, A. Synchronization and Propagation of Vortex-Induced Vibrations in Francis Turbines due to Lock-In Effects Based on Coupled Vibro-Acoustic Mode Shapes. Proceedings of the 4th International Meeting on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Belgrade, 2011.

PUMPED

Simulation-driven development increases pump efficiency and reduces product development time at one of the world's leading pump manufacturers.

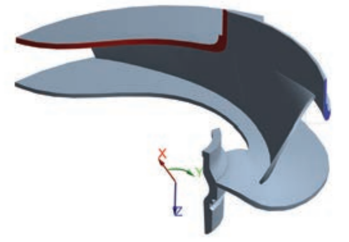
By Jakob Vernersen, Head of Mechanical Technology,
and **Nicholas Engen Pedersen**, Chief Hydraulic Engineer, Grundfos, Bjerringbro, Denmark

Grundfos calculates that pumps currently account for 10 percent of the world's total electricity consumption. The company's engineers continually focus on optimizing pump efficiency to reduce energy consumption and operating expenses while having a positive impact on the environment.

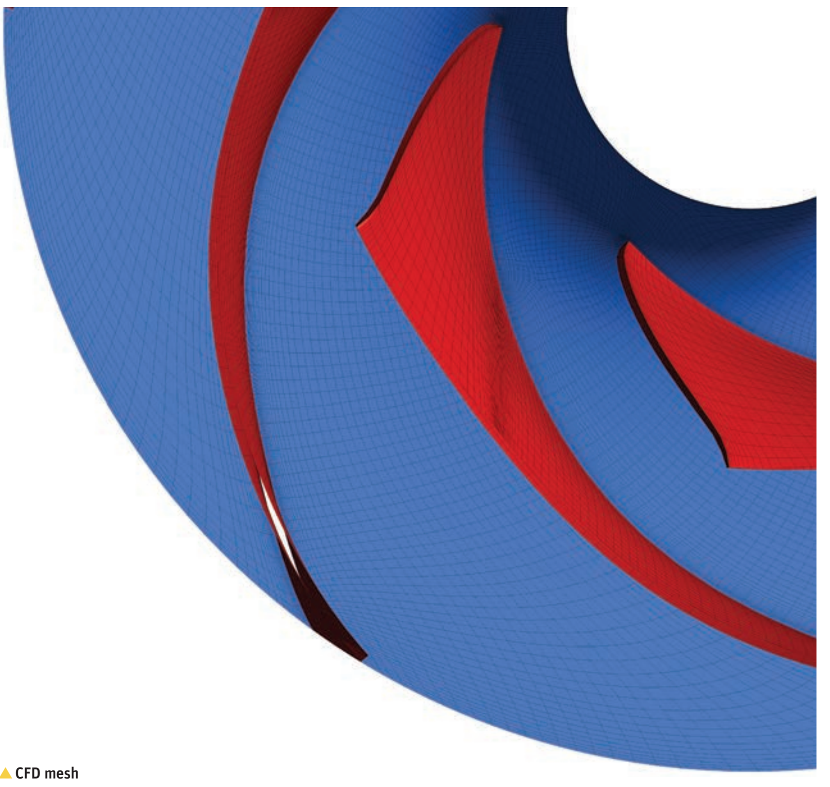
To design pumps, Grundfos has been using finite element analysis (FEA) since the 1980s and computational fluid dynamics (CFD) since the 1990s. Originally, FEA and CFD were

With simulation, Grundfos engineers significantly improved hydraulic efficiency of the new pump.

Parametric model of the impeller used to define hydraulic geometry



To reduce the size of the problem and obtain acceptable mesh density, cyclic symmetry was used for the impeller.



CFD mesh

Using Simulation-Driven Product Development to Increase Pump Efficiency and Reduce Development Time at Grundfos

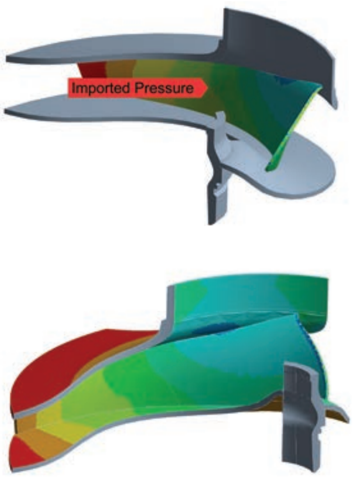
used for research and troubleshooting. Some of the information from these simulations was helpful in new product design. Then, more than 15 years ago, Grundfos chose to use simulation for product design to develop reliable and efficient products. Most recently, the company linked together a chain of simulation tools into an automated design loop, called PumpIt, that enables engineers to investigate hundreds of designs without manual intervention. An optimization routine explores the entire design space and identifies optimal designs based on criteria set by Grundfos engineers. PumpIt provides a user interface that allows engineers to specify the type of component they want to design, such as an impeller, guide vane or volute. Then PumpIt initiates an automated design optimization loop that calls simulation tools, including ANSYS CFD software, to explore the design space.

In a recent example, Grundfos engineers used PumpIt to drive ANSYS CFX CFD simulation to optimize the hydraulic surfaces of a new pump design. They increased the hydraulic efficiency of the pump by 1 percent to 2 percent while extending the maximum efficiency level over a wider range of flow rates. The team used ANSYS Mechanical FEA software to optimize the pump from a structural standpoint, ensuring that it will meet fatigue life targets while minimizing weight and manufacturing cost.

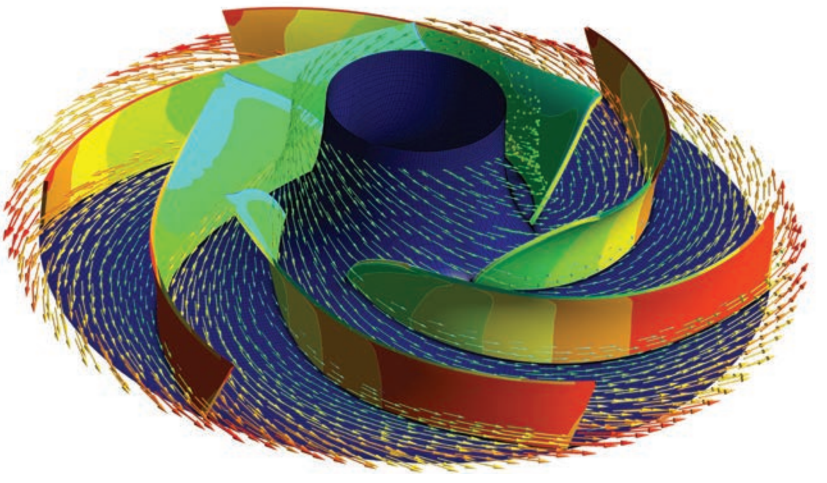
PUMP DESIGN CHALLENGES

With an annual production of more than 16 million pumps, Grundfos is one of the world's leading pump manufacturers. The company is the leading producer of circulator pumps used for heating, ventilation and air conditioning in private houses, offices and hotels, with a market share of approximately 50 percent. Grundfos also produces centrifugal

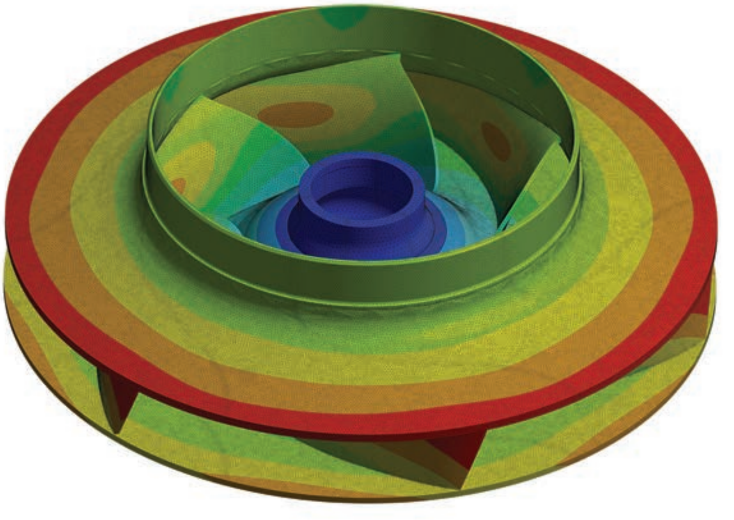
The multiple physics simulation process reduced overall design time for the new pump design by 30 percent and saved approximately $400,000 in physical prototyping.



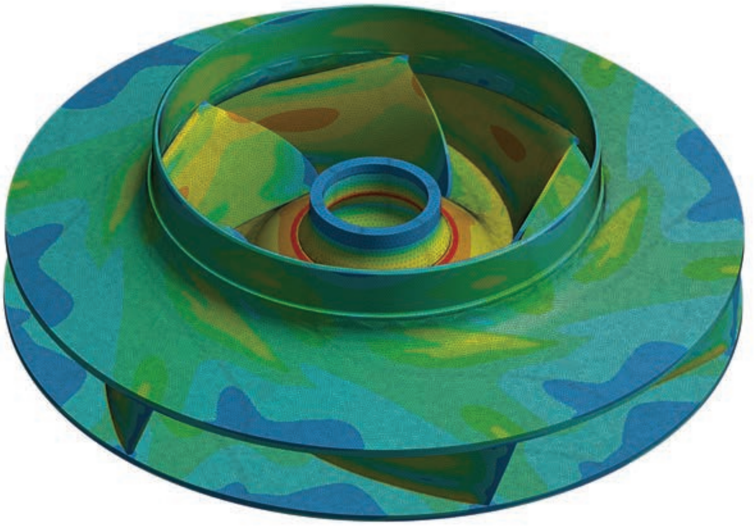
CFD pressure results were mapped onto structural simulation as loads.



CFD results demonstrated using pressure contours and velocity vectors.



Deformation of the hydraulic impeller



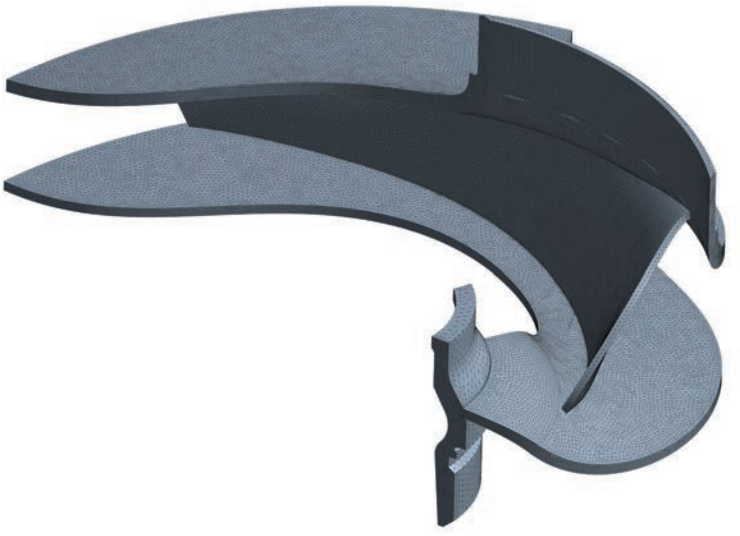
Global stress simulation

pumps for water supply, sewage, boiler, pressure boosting and other industrial applications, as well as pumps that are integrated into other original-equipment manufacturers' products.

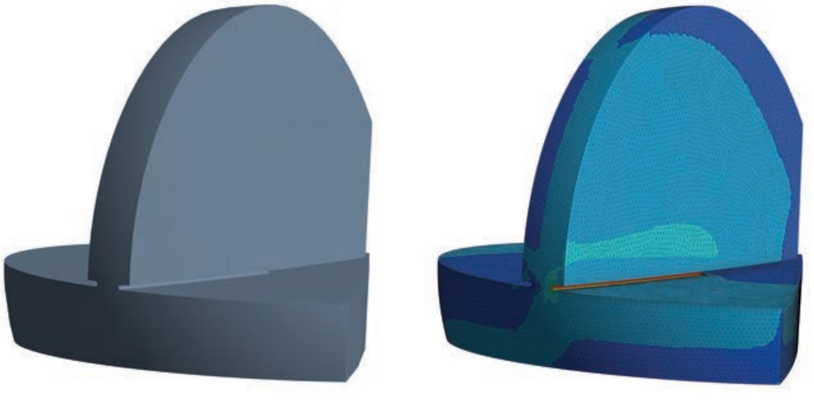
To optimize Grundfos equipment, engineers must improve the peak efficiency of the pump. Delivering a relatively flat performance curve with high efficiency levels over as wide a range of flow rates as possible is another goal. The flow rate of the pump depends on the installation; a relatively flat performance curve can deliver high levels of efficiency for many applications. A flat performance curve also reduces cavitation, thereby increasing longevity. Another important goal in pump design is to meet component structural requirements with the minimum amount of material. Minimizing material usage reduces manufacturing costs and decreases component weight. Lighter pump components make it possible to use less-expensive bearings while reducing noise and vibration.

FLOW SIMULATION

Recently, Grundfos engineers used PumpIt to design multi-stage hydraulics for a new pump. The team developed parametric models of all pump components to define hydraulic geometries of the surfaces in contact with the fluid being pumped. The design objectives were to maximize hydraulic efficiency and deliver maximum efficiency over the widest possible range of flow rates. The



▲ Structural mesh



▲ Grundfos engineers used submodeling to evaluate critical areas of the component with a high level of detail without greatly increasing simulation time. Image on right reveals stress results.

PumpIt tool used design of experiments (DOE) to create a series of iterations that explored the design space for each component. PumpIt then generated the geometry for each design iteration and issued a call to the CFD software to simulate each iteration.

The initial DOE included about 40 designs. It determined which parameters played the most important roles in the simulation, and it delineated broad ranges of the most promising values for these parameters. The parameters and values were then used as the starting point for a Kriging-based optimization algorithm that automatically generated additional design iterations for CFD evaluation. The optimization routine evaluated the results of each design iteration and then performed additional iterations based on these results. Each iteration moved the design closer to the efficiency objectives.

The initial analyses were performed with a coarse-density mesh and standard turbulence model to reduce the time required to obtain promising parameter values. As the design converged on optimal values, the CFD model was refined by applying a finer mesh and a more-advanced turbulence model. To obtain high-fidelity results for an optimized design, engineers used up to 48 cores to perform analyses on a high-performance computing cluster. The cluster had more than 1,000 cores using more than 8 terabytes of random access memory and 50 terabytes of high-speed storage running on the Lustre® parallel file system. Engineers simulated hundreds of design iterations overnight.

The next step was to evaluate the manufacturability of the most promising designs. Grundfos engineers considered how easily each design could be produced with several alternative production technologies. To do this, they evaluated the geometric parameters that are required for each production method. In this case, they decided to stamp the components from stainless steel sheet metal. The performance statistics for the best designs were displayed in the PumpIt multidimensional solution visualizer that can be configured to display any outcome variable.

Grundfos was able to simulate hundreds of design iterations overnight.

STRUCTURAL ANALYSIS

After optimizing the hydraulic design with CFD, Grundfos engineers performed structural analysis with ANSYS Mechanical to ensure that each component would meet fatigue life requirements while keeping cost and weight as low as possible. They mapped the hydraulic pressure determined by CFD simulation onto the finite element analysis to specify loading conditions that accurately match the complex hydraulic pressure distribution. To minimize computational cost and accommodate an acceptable mesh density, cyclic symmetry was applied.

The welding operation on the sheet metal components was modeled according to the recommendations of the International Institute of Welding publication's "Recommendations for the Fatigue Assessment of Welded Structures by Notch Stress Analysis." Fatigue life was evaluated using the notch stress method. The team also used submodeling to evaluate critical areas of the component with a high level of detail without greatly increasing simulation time.

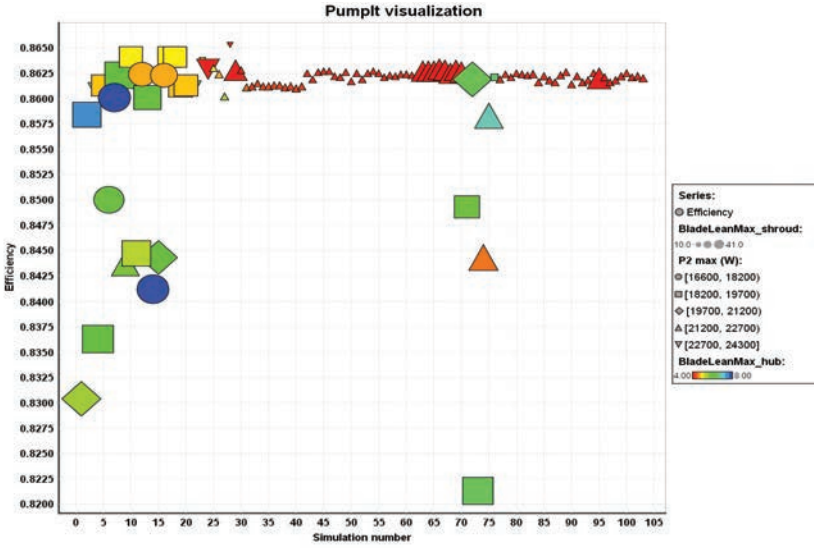
Engineers configured the automated workflow to vary the welding parameters so they could determine how to optimize the welding process. The team also used structural simulation to predict the potential for failure in any given lifespan. The only other way to obtain this information would have been with an expensive and lengthy physical testing program. Grundfos engineers performed several sensitivity studies on input variables, including welding thickness, air gap and flow points, to determine the robustness of the design with respect to fatigue life of components. By providing a statistical distribution of fatigue life for each input variable, simulation made it possible to improve component quality and reliability.

On other structural components, topology optimization was used at the concept level of the design process to arrive at a design proposal that was then fine-tuned. This replaced time-consuming and costly design iterations, thus reducing development time and overall cost while improving design performance. Dassault Systèmes' Tosca topology optimization software was coupled with the ANSYS Workbench interface and the ANSYS structural solver.

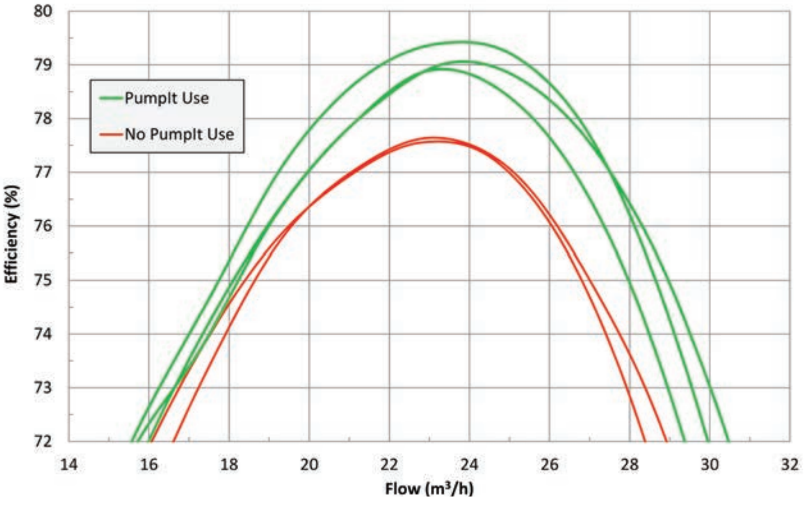
PERFORMANCE AND TIME TO MARKET IMPROVED

The result was a substantial improvement in product performance with reduced design cost and lead time. With simulation, Grundfos engineers significantly improved the hydraulic efficiency of the new pump. Compared to a traditional prototype-based design process, the multiple physics simulation process reduced overall design time for the new pump design by 30 percent and saved approximately $400,000 in physical prototyping.

Recently, Grundfos made the decision to take another step forward and migrate its simulation process to the ANSYS Workbench environment. Workbench was selected as the integration framework because of its ability to seamlessly integrate the broad portfolio of ANSYS applications as well as third-party applications. ANSYS Workbench can be used to develop state-of-the-art user interfaces, workflow and applications. Workbench's common tools and services, including parameter management, units and expressions, application development tools, and solver coupling capabilities, can deliver considerable time savings. ANSYS HPC Parametric Pack licenses make it cost-effective for Grundfos to simulate large-scale parametric design variations. Using Workbench workflows, engineers are creating a simulation-driven development process for multiple physics to design hydraulic components such as impellers, guide vanes and volutes. This tool could allow Grundfos to raise pump performance to an even higher level while providing further reductions in design cost and lead time.



The ability of each iteration to meet design objectives is graphically displayed in the PumpIt solution visualizer.



Grundfos engineers made a substantial improvement in pump hydraulic performance.

Simulation made it possible to improve component quality and reliability.

Grundfos is supported by ANSYS channel partner EDR Medeso, which has supported Grundfos during creation and implementation of a simulation-driven development strategy.

RISING TIDE

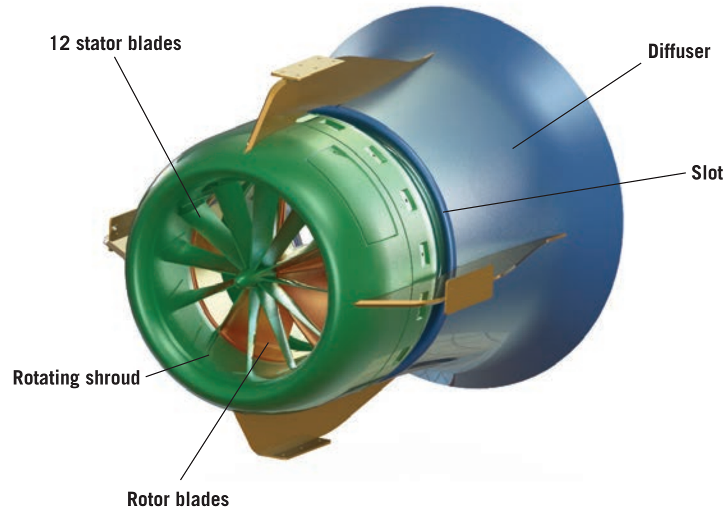
By employing simulation, a consulting firm optimizes the design of an innovative tidal current power generator to produce four times as much power as earlier designs.

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

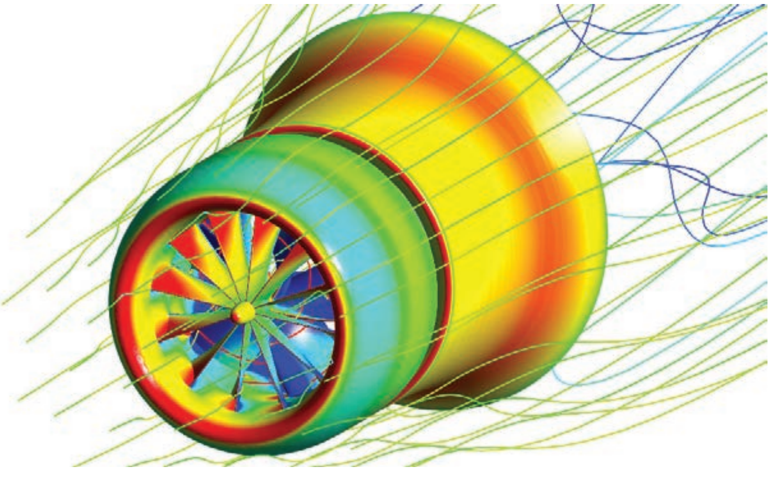
Addressing growing worldwide electricity consumption and the desire to reduce greenhouse gas emission from fossil-fuel–based resources drives the demand for renewable energy technologies. Along with harnessing wind and solar power, capturing the vast kinetic power in the world's tidal currents, ocean 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

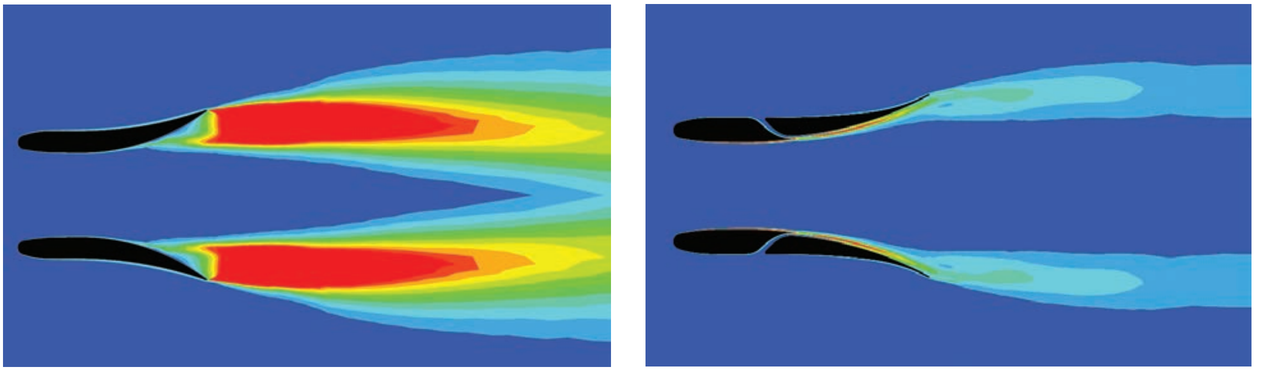
Urch basically developed this unique design concept on a napkin, but realized that it was essential to optimize the design, validate the concept and provide an estimate of how much power it would generate before taking the next step. The traditional approach would have been to build a series of prototypes 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

TURBOMACHINERY SIMULATION PRODUCTIVITY ADVANCES WITH ANSYS



▲ 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.

GENERATING INNOVATION

A world leader in small hydropower systems and engine cooling pumps, Gilkes built a long history of success based on proven products. In 2013, executives recognized that, to maintain leadership, product innovation was needed. By building in-house expertise in engineering simulation, the company is re-inventing its product line, both quickly and cost-effectively. This successful 162-year-old business has a few lessons for other companies targeting major innovation.

By *ANSYS Advantage* Staff

Increasing environmental regulations, coupled with government incentives for green products and systems, have created dramatic changes for engineering teams in many industries. For example, in the power generation industry, more plants are being constructed using wind, water and solar power, which has created a fast-growing market and a new set of customer needs. Product development teams focusing on engines or engine components face stricter emissions standards as well as weight restrictions that support greater fuel efficiency.

Gilbert Gilkes & Gordon Ltd. — commonly referred to as Gilkes — is a leading manufacturer serving both the power generation and engine industries. Gilkes Hydro is a global leader in hydropower systems that generate electricity from water, with more than 6,700 turbines installed in more than 80 countries. Gilkes Pumping Systems manufactures a range of sophisticated pumps for the cooling of high-horse power diesel engines, supplying many of the world's top diesel engine manufacturers.

Founded in the United Kingdom's Lake District in 1853, Gilkes is steeped in tradition. Its main factory has been in the same location since 1856, and it has been under the same basic ownership since 1881. With a loyal customer base and a stable of proven product designs, Gilkes was able to lead the global market in small hydropower systems and engine cooling pumps for many years.

"The traditional approach at Gilkes was to create a high-performing design through testing, optimize it for production in our factory, then rely on variations of that design for years," said Lindsey Entwistle, mechanical design engineer for cooling pumps at Gilkes. "Customers were very happy with the product's performance, and it was an approach that worked for many years."

Gilkes had to be able to guarantee higher turbine performance if it were to retain its market share.

Alan Robinson, Jo Scott and Lindsey Entwistle

However, in the last decade, the landscape began to change in both industries that Gilkes serves. Due in part to government subsidies for renewable energy, the market for hydro turbines began to grow quickly in many regions of the world, new competitors appeared and Gilkes had to guarantee higher turbine performance to retain its market share.

In addition, increasing environmental awareness meant new regulatory standards for diesel engines. Gilkes' existing pump designs required higher levels of performance to contribute to decreased emissions and other environmental goals, as well as reduced production costs. For the first time in years, Gilkes' product requirements were dramatically changing.

SIMULATION: A COMPETITIVE ADVANTAGE

Across both market segments, Gilkes' historic competitors moved quickly to develop innovative designs that capitalized on these opportunities — relying heavily on computational fluid dynamics (CFD) simulation tools to drive fast design and market launch of new products that answered these needs.

Engineering simulation was not a new concept to Gilkes. Historically, when design analysis was needed, the company had outsourced CFD simulation to experienced consultants. Gilkes had also sponsored the work of a Ph.D. student, at nearby Lancaster University, who had built his thesis around answering one of Gilkes' pressing engineering challenges.

"We had dabbled in engineering simulation in the past, but not really committed to it as a central strategy to support product innovation," explained Alan Robinson, research and development manager for Gilkes hydroturbines. "We have a history of under-promising and over-delivering. But with efficiency guarantees so heavily weighted in bid evaluations, we knew we had to improve our product performance — we had to innovate so we could offer higher guarantees and keep our valued ethos. And the engineering team responded with a proposal to create an in-house simulation capability — because we recognized that simulation had become a key competitive edge we were lacking."

"We showed the board of directors the capabilities of simulation software and how it could help us quickly redesign our products," Robinson continued. "The board agreed to make a significant investment in not only technology but new engineering staff with simulation skills."

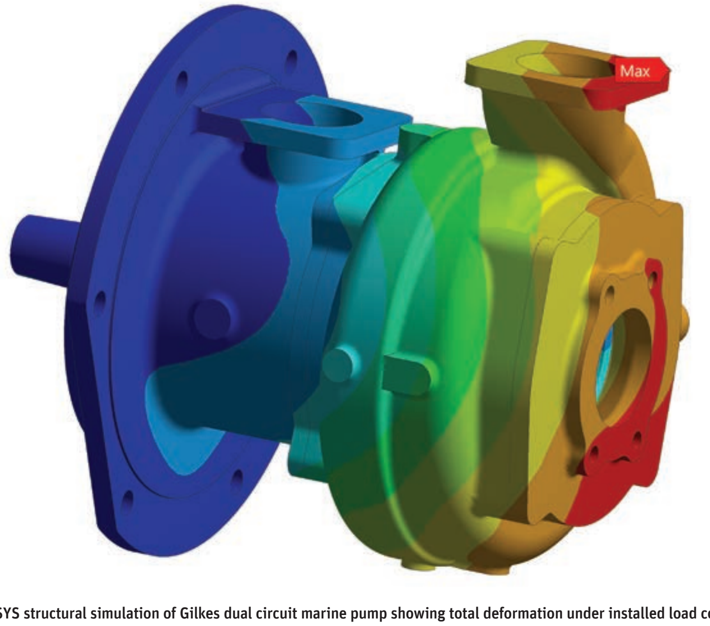
A NEW CAPABILITY TAKES SHAPE

In 2013, Jo Scott was hired as an experienced CFD engineer for Gilkes' hydro turbines business. Because Scott had used simulation software for 20 years in his previous positions, he became the champion of simulation within both Gilkes business units.

"Our first lesson was to choose the software carefully," noted Scott. "Even after we decided to purchase a best-of-breed software, we had to select the actual solutions. We realized that CFD simulation was a requirement for both businesses, but that the pumps engineering team also needed to do finite element analysis (FEA) to ensure structural robustness. But there were many levels of FEA analysis tools, so

Gilkes has a long history of engineering excellence.

Gilkes cooling pumps



ANSYS structural simulation of Gilkes dual circuit marine pump showing total deformation under installed load conditions

Pelton runners on Gilkes shop floor

we had to match the solution to our day-to-day challenges."

While Scott initially tried to train some of his colleagues in simulation software, he quickly realized that the best strategy was to leverage the expert training provided by ANSYS. "Even though I knew the CFD software very well, it simply wasn't time- and cost-efficient to have me manage the internal training — and I had little working knowledge of FEA solutions," said Scott. "So we had a team of people attend formal software training, which helped us get a core group of users up and running."

Today, Scott is joined by three part-time ANSYS users in the hydro turbines business. In the cooling pumps business, Gilkes has four engineers using CFD software and another three team members using FEA software to analyze structural issues.

Throughout, Gilkes has made full use of phone-based support and an online customer portal to get answers to technical questions. "Software providers offer web- and phone-based support for a reason — and you shouldn't be shy about using those resources," stated Scott. "There's so much product knowledge there."

While Gilkes began with a single seat of software — relying on a leasing approach for additional seats — in 2015 the company realized that it needed to make a longer-term commitment. "Once we were able to assess the real usage of simulation software at Gilkes, we saw that it made more sense to buy licenses instead of leasing them," said Scott. "It was a financial decision based on how frequently simulation was being used by our team by 2015."

We showed the board of directors the capabilities of simulation software and how it could help us quickly redesign our products. The board agreed to make a significant investment.

A WELCOME CHANGE

At Gilkes, the adoption of engineering simulation was embraced by many existing employees who were eager to learn leading-edge skills. "Our engineers had been doing a lot of complex calculations and design work using more-traditional methods, so they were extremely enthusiastic about having new software do the work for them," said Robinson. "They wanted to get up to speed on the latest practices."

The new focus on simulation is also attractive to recent graduates who are ready to apply the skills they have learned in college. "Traditionally, there was a gap between how Gilkes engineers were working and the way new engineers were being trained at university," noted Entwistle, who joined Gilkes in 2014. "But that gap was disappearing by the time I arrived. And today, Gilkes really is at the forefront of engineering practices. It's exciting to work here."

While the change was welcome, Robinson noted that it was challenging from a cultural standpoint. "Previously, we had engineers spreading their skills

We have won some sizable customer contracts because we can produce innovative designs more quickly and cost-effectively.

thinly to oversee entire projects," he explained. "Now we've installed a modular process in which people have different roles and different areas of expertise. We have specialists at every stage, including our CFD and FEA experts. We've had to change our process and re-align employees' roles, but that was necessary to fully adopt simulation as a core competency."

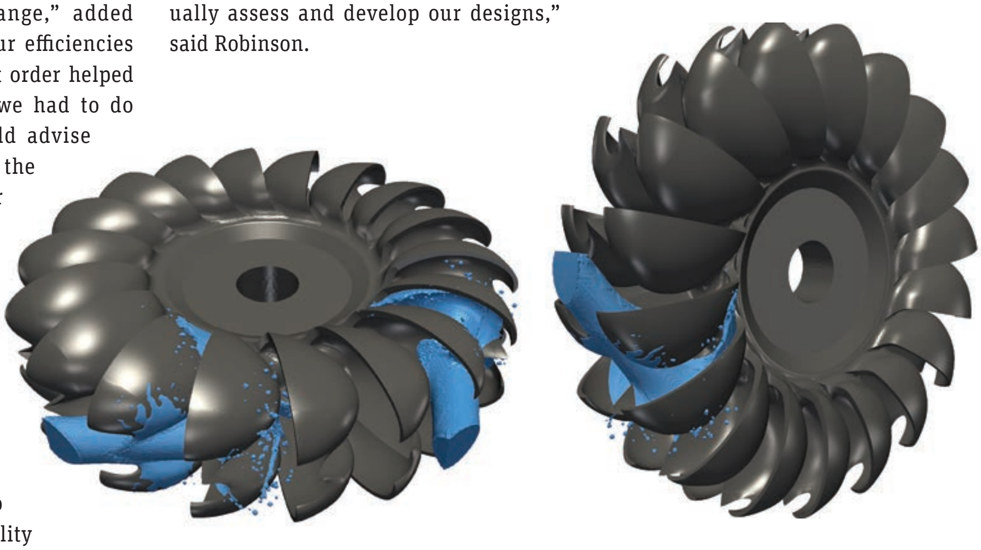
"It was helpful that everyone recognized the need to change," added Robinson. "We realized our efficiencies had to improve, and a lost order helped everyone recognize that we had to do things differently. I would advise other businesses to share the top-level vision with their engineers, because that certainly helped us overcome any cultural resistance at Gilkes."

LAUNCHING A NEW ERA

In just three short years, Gilkes has transformed from having no internal simulation capability to having 10 engineers regularly using simulation software. The company has invested approximately £150,000 in building this capability — including software licensing, hardware and training. The company is now looking into the creation of a high-performance computing (HPC) cluster to manage large simulations and make its analysis capabilities even more powerful.

"Engineering simulation now forms the basis of a strategy of analysis that is being used to promote intelligent, blue-sky design thinking, where we continually assess and develop our designs," said Robinson.

"It's hard to measure the financial impact of our investment in simulation," he continued. "But I can tell you that we are now seeing a return on that investment due to winning more contracts because we can produce innovative designs more quickly and cost-effectively. We believe that simulation has made a real difference already — and that it's positioning Gilkes for a new era of success."

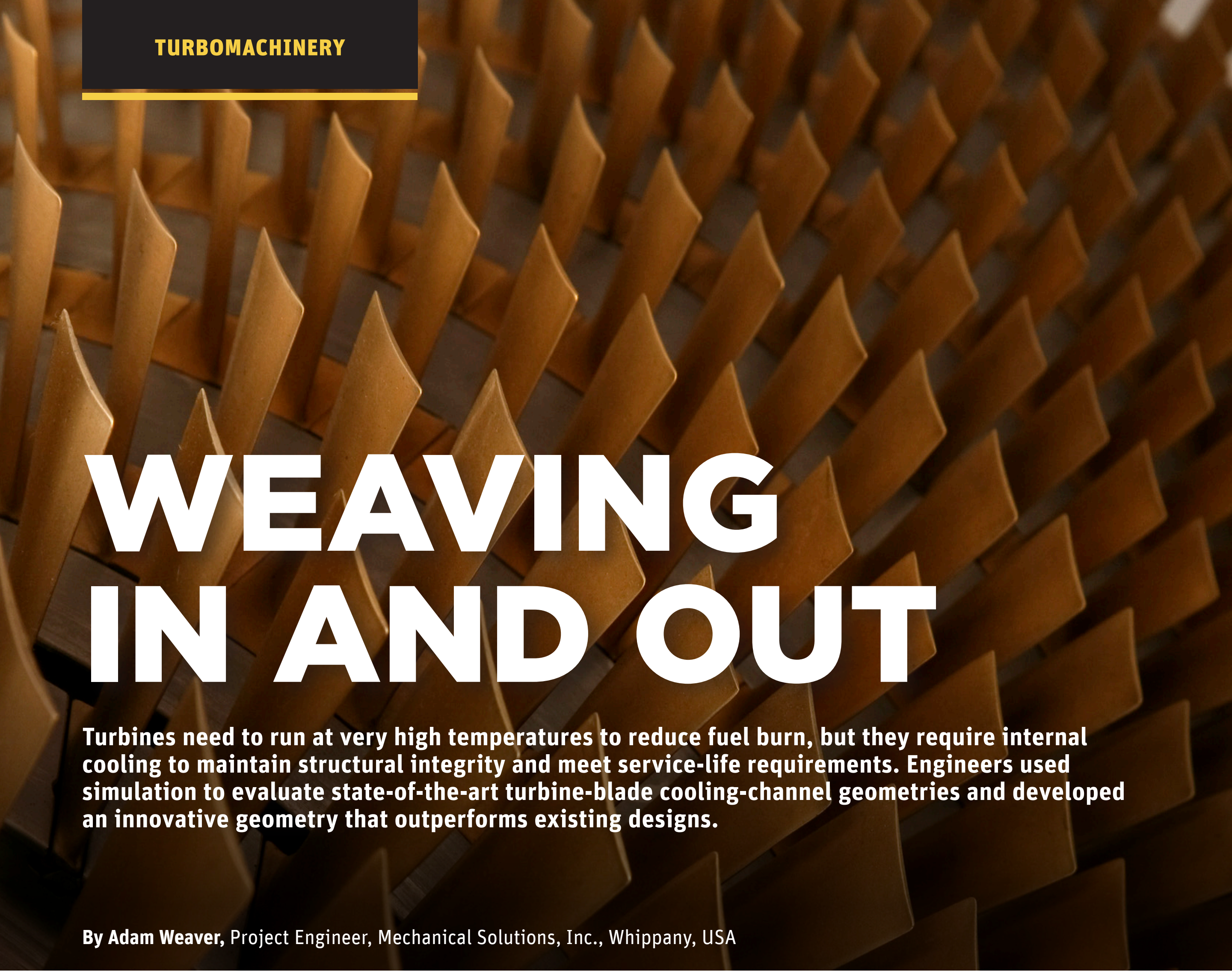


ANSYS CFD simulation of Pelton turbine runner

New to Simulation? Four Lessons from Gilkes

- **Choose the software carefully.** Make sure that the provider is best in class, but also choose individual solutions that meet your daily engineering challenges.
- **Capitalize on the provider's knowledge base.** Expert training, phone-based support and web support are there to help customers. Take advantage of these resources.
- **Structure licensing around your actual use of the software.** Gilkes first leased software, then bought the right number of licenses after usage was fully understood.
- **Communicate the need for the change.** Even positive changes can be hard to accept unless employees understand the reasons why they need to work differently.

GILKES



WEAVING IN AND OUT

Turbines need to run at very high temperatures to reduce fuel burn, but they require internal cooling to maintain structural integrity and meet service-life requirements. Engineers used simulation to evaluate state-of-the-art turbine-blade cooling-channel geometries and developed an innovative geometry that outperforms existing designs.

By Adam Weaver, Project Engineer, Mechanical Solutions, Inc., Whippany, USA

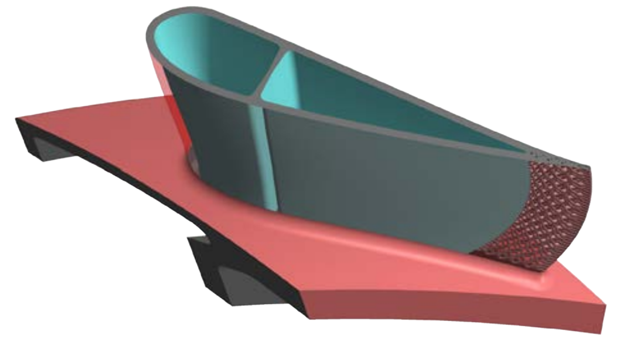
Natural gas–fueled turbines operate more efficiently at high temperature. Gas temperatures in the latest generation of turbines can reach 2,700 F, but blade materials can typically only withstand temperatures of about 1,700 F, so cooling is required. However, because the trailing edges of turbine-blade airfoils are very thin, there is little opportunity to cool them without endangering the structural integrity of the blade. Another challenge is that the amount of flow required to cool the trailing edge, as well as pressure losses within the cooling circuit, must both be minimized because the power required to generate this flow detracts from the efficiency of the engine.

Trailing edges with internal cooling channels are usually manufactured using investment casting, in which ceramic cores are used to create internal channels. This has limited designers to developing cooling channels with very simple geometries. However, new manufacturing technologies can now create more complex cooling channel shapes. But what shape is the best? The author and his colleagues at Purdue University used ANSYS CFX to investigate the performance of some new geometries and to create an innovative shape that delivers significantly higher performance.

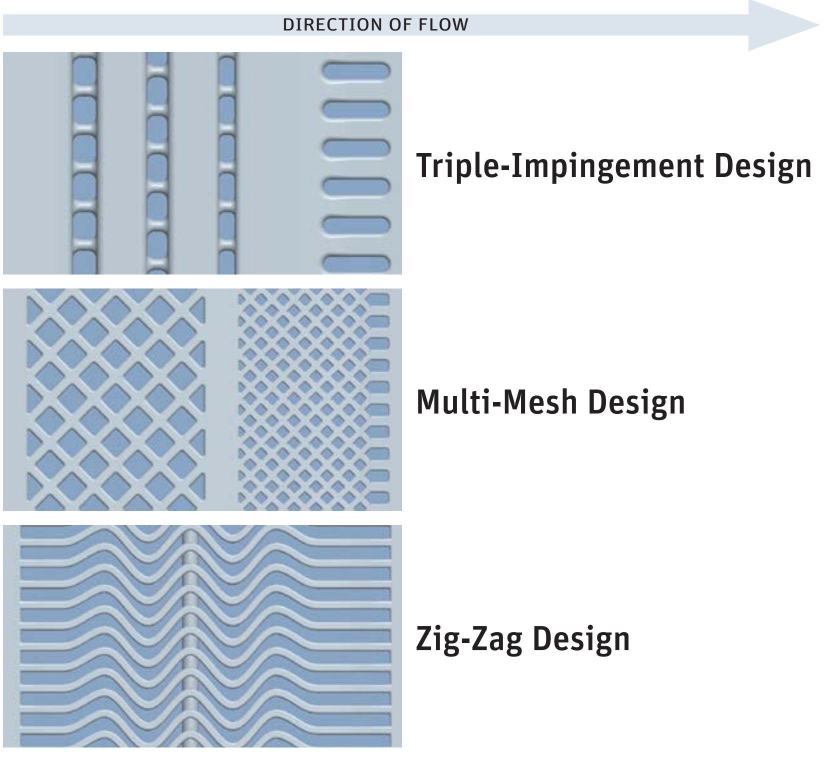
PRODUCING CERAMIC CORES FOR COOLING CHANNELS

Investment casting has been the technology of choice for producing gas turbine components with complex airfoil shapes and internal cooling passage geometries. For this type of casting, the lost wax process creates a ceramic shell with an interior that corresponds to the airfoil shape. One or more ceramic cores are positioned within the shell to form the internal cooling passages. Molten alloy is introduced into the shell and allowed to cool and harden. Then the shell and core are removed by mechanical or chemical means to leave the finished component. Until recently, design choices for cooling

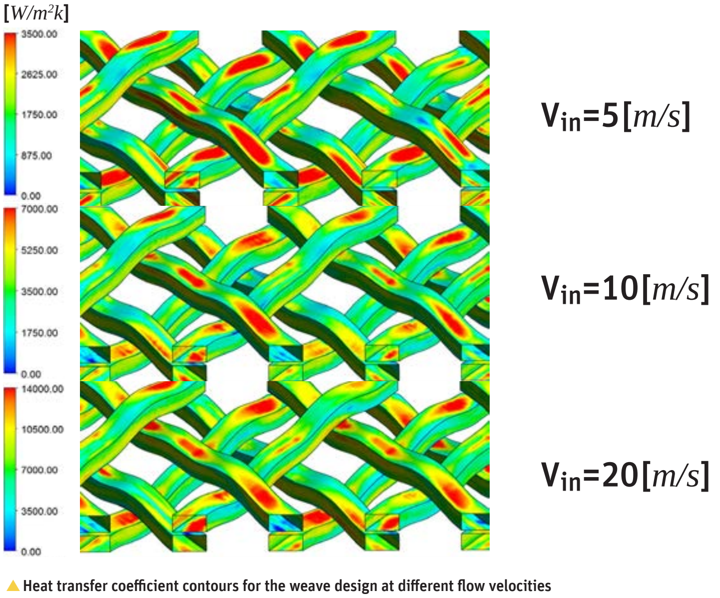
Because trailing edges of turbine-blade airfoils are very thin, there is little opportunity to cool them without endangering the structural integrity of the blade.



Position of new channels within the blade



Three new cooling channel geometries enabled by new manufacturing methods



Heat transfer coefficient contours for the weave design at different flow velocities

channels have been limited by manufacturers' abilities to produce ceramic cores with geometric features of about 1 or 2 mm. So only very simple cooling channel shapes could be used for turbine blade cooling passages.

These simple channel geometries are not very efficient at transferring heat from the surrounding metal. If the cooling channel is straight with a square cross-section, for example, a hot thermal boundary layer will form near the solid surface, but the flow in the center of the channel will absorb little heat. Much of the energy expended to cool the airfoil is wasted.

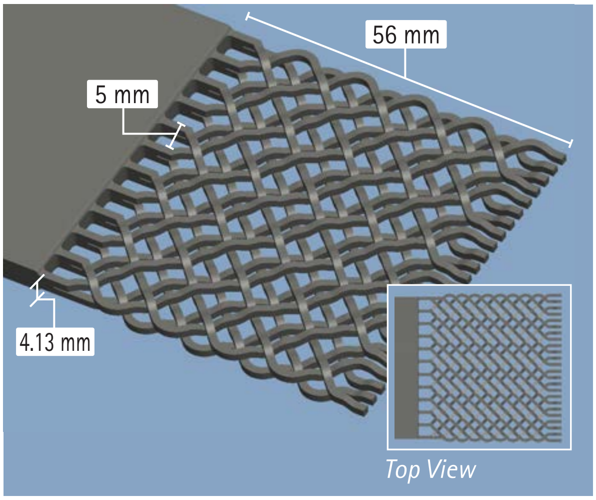
New manufacturing methods introduced in the last couple of years have broadened the scope for possible cooling channel geometries. One new manufacturing approach involves slicing a computer model of the cooling channels into slivers about 25 microns thick, which are used to create photomasks. The photomasks are in turn used to etch metal foils that are laminated together to make an extremely accurate 3-D cavity. The master pattern produces a mold from a material such as silicone, which can be used to cast ceramic material to very high levels of accuracy.

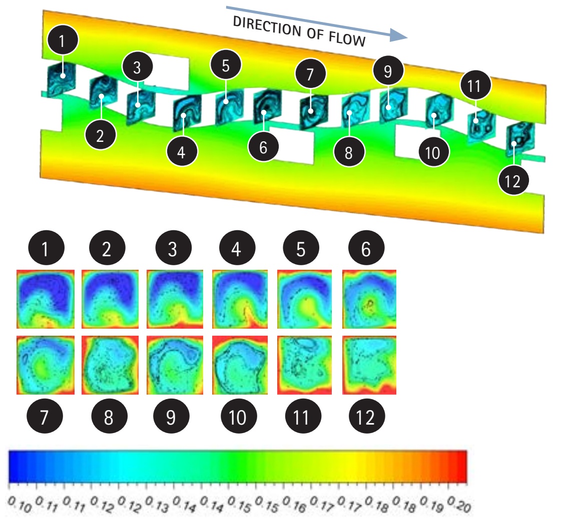
SIMULATING NEW COOLING CHANNEL GEOMETRIES

These new manufacturing methods led the team at Purdue to explore several cooling channels with more complex geometries that can break up the boundary layer and increase heat transfer.

- The triple-impingement design causes high-velocity jets to form through area contractions and directs the jets at perpendicular solid surfaces.
- The multi-mesh design uses square posts angled at 45 degrees to the flow direction to deflect and mix the flow with jet-to-jet interactions.
- The zig-zag design constricts the flow to channels and bends these channels to cause secondary flow structures and drives cool fluid to the walls.

FLUID STRUCTURE INTERACTION MAKES FOR COOL GAS TURBINE BLADES — ARTICLE
ansys.com/weaving101





▲ The weave design has a flow divider at the upstream end and aligned ducts at the trailing edge of the airfoil.

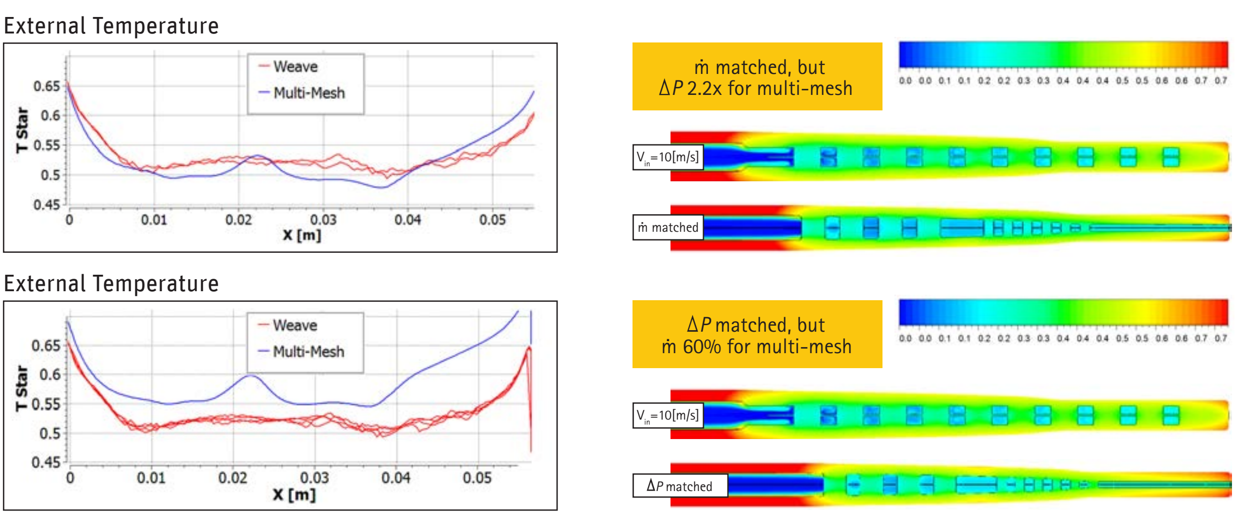
▲ Streamlines and temperature contours projected on cross sections of the weave cooling channel show how it breaks up boundary layers.

The Purdue team used CFD to evaluate the performance of these designs along with the author's own design, which has two layers of 45-degree–angled channels that undulate at intersections to avoid contact and constantly disrupt the flow direction. The team performed conjugate heat transfer analysis with the shear-stress transport (SST) turbulence model for all four designs. The temperature of the hot gas surrounding the trailing edge was maintained at 1,755 K, and the external heat transfer coefficient was set at 2,000 W/m^{2}K. The temperature of the cooling flow at the inlet passage was 673 K, and the pressure at the exit of the duct was maintained at 25 bars. Inlet coolant mass flow or pressure was varied to investigate a range of performance. The passages were 2 mm tall.

ANSYS meshing and fluids capabilities allowed the team to create models that consistently converged to a solution with node spacing as small as 0.001 mm, although more commonly the spacing was 0.01 mm near the wall where mesh density is most critical. Other codes that were tried in this application often did not converge at this node spacing. The researchers validated the CFD results against a test problem consisting of a planar jet impinging on a flat plate and achieved very good correlation.

IDENTIFYING THE BEST GEOMETRY

The researchers simulated the three existing designs to select equivalent points on the performance map so that they could directly compare any two designs. The results show that, for a given mass flow rate, the multi-mesh design performs about 5 percent better than zig-zag and 10 percent better than triple-impingement in terms of the amount of heat transferred. For a given pressure drop, zig-zag provides about 30 percent more heat transfer than triple-impingement and multi-mesh, but requires about twice the cooling flow rate compared to



▲ Temperature distribution using matched mass flow (top) and matched pressure drop (bottom) shows the superiority of the weave design over the multi-mesh design.

The weave design produces a higher flow rate of cool gas and thus provides 20 percent greater heat transfer, resulting in a relatively cooler blade.

multi-mesh and triple-impingement methods. The researchers concluded that multi-mesh is the best design among these three because it achieves maximum cooling with minimum cooling fluid flow.

The researchers then compared the multi-mesh design to the new weave design. With the same mass flow rate, multi-mesh performs similarly in terms of heat transfer, and even provides locations where external temperatures are lower than those of the weave design. However, multi-mesh's strenuous flow path resulted in more than two times the amount of pressure loss when compared with the weave design, demonstrating a much less efficient use of the cooling airflow. When pressure drop is matched between the two designs, the weave design produces a higher flow rate of coolant gas and thus provides 20 percent greater heat transfer, resulting in a relatively cooler blade.

The increase in heat transfer makes it possible to operate at a higher combustion temperature while maintaining the same blade temperature, thereby providing a higher thermal efficiency cycle for the engine. The amount of efficiency that can be gained will depend greatly on the specific engine used and the owner's preferred operating limits. The Air Force has awarded a Small Business Innovation Research (SBIR) grant to integrate the weave design into the Air Force Research Laboratory's research turbine at Wright-Patterson Air Force Base with the goal of ramping up to full-scale testing of the new design in 2016.

References

[1] Weaver, A.M.; Liu, J.; Shih T., I-P. A Weave Design for Trailing-Edge Cooling. The American Institute of Aeronautics and Astronautics (AIAA) SciTech Forum and Exposition. 2015.

[2] Weaver, A.M.; Liu, J.; Shih T., I-P; Klinger, J.; Heneveld, B.; Ames, R.; Dennis, R.A. Conjugate CFD of Three Trailing-Edge Cooling Designs. *Proceedings of ASME Turbo Expo 2013 Power for Land, Sea and Air.* 2013.

Keeping Turbines Cool: A Multi-faceted Challenge

The high and increasing operating temperatures of contemporary gas turbines present "hot section" engineers (including those who work with high-pressure turbines) with many challenges. Both materials and fluids engineers must contribute to overcoming heat transfer challenges. One part of the fluids solution is predicting the flow in the complex passages inside blades; a related concern is understanding the main flow path physics.

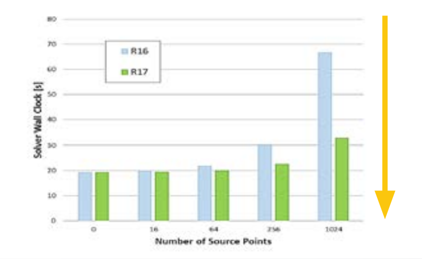
Much of the cooling air supplied by the compressor is ultimately delivered to the main flow path, providing a thermal cushion between the hot main flow and the metal surfaces that enclose that flow. The flow field is quite complicated because the interaction of the cooling jets and the main flow results in flow features that are much smaller than the main flow path scales. Thus, a full and detailed computational resolution is still a challenge. In addition, the flow is transient due to the relative motion between the rotor and stator blades.

One useful simplifying approach is to model the cooling jets as sources of mass and energy. Such sources could number in the hundreds. In the most recent release of ANSYS CFX, the workflow is streamlined so that any additional computational cost for inclusion of hundreds or thousands of source points is minimized.

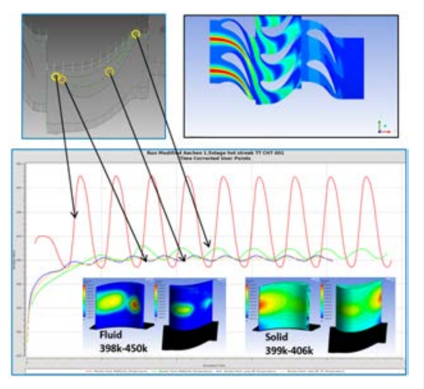
Engineers would also like to simultaneously solve for the heat transfer in blades and in the flow path. Again a problem of scale complicates matters, because the time scale for conduction is several orders of magnitude different than in the fluid. ANSYS provides a solution for transient blade row simulations to overcome this taxing problem.

With each release, ANSYS works to improve the fidelity, efficiency and scalability of its turbomachinery tools.

— **Brad Hutchinson,** Global Industry Director, Industrial Equipment and Rotating Machinery, ANSYS



A new capability reduces total CPU time by as much as 70 percent when a large number of source points are involved.



A transient blade row simulation with conjugate heat transfer shows the effect of thermal hot streaks and their temporal variation.

Powered by Innovation

Known for game-changing product innovations, Pratt & Whitney has relied on engineering simulation to fuel its design process for over 35 years. Al Brockett, former vice president of engineering module centers, discusses the role of robust design in delivering revolutionary new products with a high degree of confidence.

PHOTO COURTESY PRATT & WHITNEY.

By *ANSYS Advantage* Staff

Simulation has been critical to our efforts to lead our industry with entirely new classes of engine designs.

Since 1925, Pratt & Whitney has been a global leader in the design, manufacture and service of aircraft engines, auxiliary and ground power units, small turbojet propulsion products, and industrial gas turbines. From its first 410-horsepower, air-cooled Wasp engine to its award-winning PurePower® engine with patented Geared Turbofan™ technology, the company continues to revolutionize engine design to anticipate changing customer needs. Pratt & Whitney's large commercial engines power more than 25 percent of the world's mainline passenger fleet. The company also provides high-performance military engines to 29 armed forces around the world.

For over three decades, Pratt & Whitney has leveraged the power of engineering simulation to launch its groundbreaking innovations with the incredibly high degree of confidence required in the aerospace and defense industry. Al Brockett, who recently retired as vice president of engineering module centers, relied on the power of simulation throughout his long career at Pratt & Whitney. Under his direction, the company's global engineering team consistently redefined what is possible via engineering simulation — making Pratt & Whitney one of the world's most sophisticated users of simulation processes and tools.

Brockett recently spoke with *ANSYS Advantage* about the changing role of simulation at the company, as well as Pratt & Whitney's increasing emphasis on robust design as a vehicle for launching its highly innovative products quickly, cost-effectively and confidently.

As a longtime advocate of engineering simulation, how have you seen its application evolve at Pratt & Whitney?

Over the course of my career, I've seen simulation transform from simple numerical calculations to the incredibly complex, multiphysics problems we're solving today. Historically, Pratt & Whitney used complex simulations only for post-design analysis and verification. But today — thanks to advances in high-performance computing, process automation and software tools — we're leveraging simulation from the earliest stages of conceptual design through detailed design, through aftermarket service, to improve both speed and fidelity of our product development efforts and management of our products in service. Simulation has been critical to our efforts to lead our industry with entirely new classes of engine designs — and these represent a true step change over traditional architectures.

In the last 15 years, we've seen the speed and power of engineering simulation improve dramatically, along with the graphic capabilities and breadth of simulation software. Those tremendous advancements allow us to visualize problems in greater detail, consider multiple physics simultaneously, and conduct simulations that consider millions of degrees of freedom, all at a pace that matches the design cycle — something we couldn't have imagined at one time.

These improvements also allow us to respond much faster to our customers' increasing demands for new approaches to engine designs that answer their pressing needs for better fuel efficiency, lighter weight and reduced emissions. With fuel costs now accounting for 45 percent of an airline's operating expenses, this is a particular concern in our industry — and simulation continues to enable Pratt & Whitney to set the industry standard in maximizing fuel efficiency.

I've seen simulation transform from simple numerical calculations to the incredibly complex, multiphysics problems we're solving today.

What role has engineering simulation played in some of your revolutionary product launches, like the new PurePower® engine?

The new products that we develop represent a multi-billion-dollar investment. Simulation helps to protect this investment by ensuring that our thousands of engineers and operations staff around the world are working efficiently, integrating functionality whenever possible, and minimizing costly rework.

In the case of the PurePower engine, we could not have developed this product, or sold it to customers, without incorporating engineering simulation. First, we needed simulation to design the Geared Turbofan technology that lies at the heart of this innovative new engine. (See sidebar "Gearing Up Performance.")

Next, we leveraged simulation to demonstrate and prove the product to our customers around the world. This engine represents a technology shift — and delivers so many huge performance benefits — that our customers were naturally skeptical. To show them the Geared Turbofan™ engine in action, it would have been necessary to build a demonstration rig, run it for thousands of hours, and transport it around the world. And simulation gave us the capability to do exactly that, in the virtual realm. When we showed our simulation results to customers, alongside physical evidence of the engine's reliability, they could not argue with the performance benefits.

As a result, we have sold five different variations of the PurePower engine to five different customers — and that leads me to the final way that simulation is helping us.

Simulation allows our engineers to move seamlessly among these five product platforms as we customize the PurePower engine design for Bombardier, Mitsubishi, Airbus, Irkut and Embraer. This is an unprecedented level of design activity at Pratt & Whitney. While we are developing five products simultaneously, they are based on a similar architecture. The teams can rapidly move from one product to another very seamlessly, and they can completely build off of one simulation to the next. We have been able to reduce the size of the overall development team needed to deliver these five product platforms, while maximizing the learning that takes place from one effort to the next.

Tell me about Pratt & Whitney's internal robust design initiative, Design for Variation.

A common approach to product design is to utilize nominal geometry with some assumed variation in material properties. This method ignores the fact that parts/products are never completely produced at nominal geometry; it leads to conservatism in margins that are built in to explain the difference between predicted capability and actual capability. Controlling variation has become one of the keys to improving performance while also improving part yield and quality. Pratt & Whitney's Design for Variation (DFV) program was created to help us improve our products by quantifying and controlling variability, uncertainty and risk. Many companies, including ANSYS, refer to this as robust design.

COURTESY PRATT & WHITNEY.

DFV is really a paradigm shift that forces our engineers to statistically analyze a broad range of product geometries, boundary conditions and materials types. The program has changed from a special initiative focused on statistical training to a high-visibility strategic priority. (See sidebar "Robust Design at Pratt & Whitney.")

The DFV concept is straightforward: If we assign a numerical value to our risks, we can manage them by making targeted changes in our designs, materials and processes that increase on-wing time for engines by managing the key sources of variation.

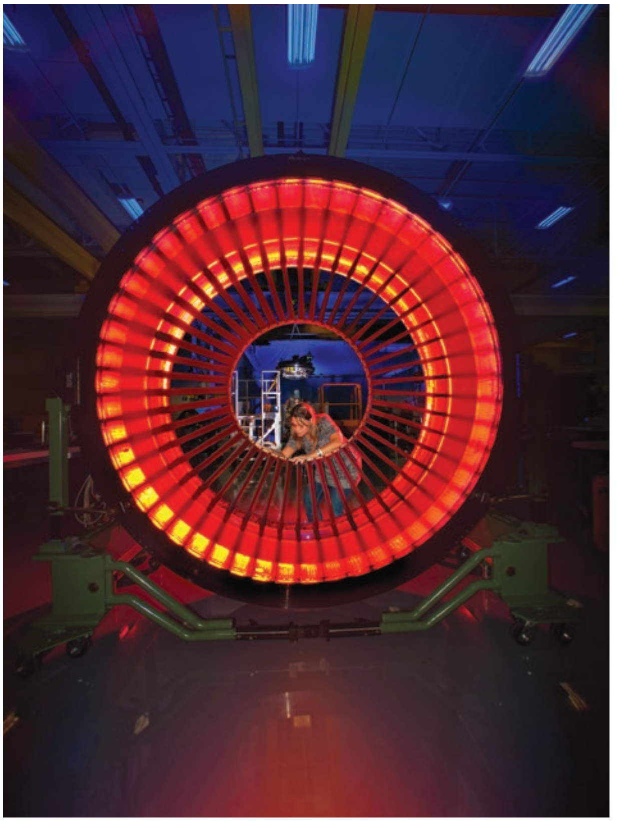
Pratt & Whitney's Design for Variation program was created to help us improve our products by quantifying and controlling variability, uncertainty and risk.

We can look at multiple physics very deeply, even assessing off-design conditions and the product system's reaction.

We examine thousands of design variations, each one slightly different, based on the probability that they will fail to meet operating requirements. We can then focus on a handful of factors that truly affect engine performance and reliability, and ignore those design points that are unimportant.

This obviously makes strategic sense, as it improves engine uptime, reduces component and maintenance costs, and protects passenger safety. But it's a massive undertaking to conduct this kind of parametric analysis.

Simulation makes DFV possible by running thousands of iterations quickly in an automated fashion. Our engineers can rapidly focus on those few design points and operating conditions that are truly critical. We can look at multiple physics very deeply, even assessing off-design conditions and the product system's reaction. The recent improvements in simulation technology are allowing us to move toward high-fidelity systems-level design, in which we will be able to isolate a dozen or so key points over an entire product system. That's exciting to consider.



COURTESY PRATT & WHITNEY.

Gearing Up Performances

Pratt & Whitney's PurePower® engine design represents one of the biggest advances in jet engines in the past 50 years. Pratt & Whitney engineers recognized that engine performance could be significantly improved if the fan and turbine that drives it could be operated at their own optimal speeds. To answer this challenge, Pratt & Whitney developed an innovative Geared Turbofan (GTF) engine design. Instead of connecting the fan directly to the low-pressure turbine via a shaft — as in conventional engine design — Pratt & Whitney engineers introduced a new reduction gearbox into the drive train.

In the resulting compact design, the bypass ratio has been improved from 5:1 to an impressive 12:1, and the low-pressure turbine develops more work in fewer stages. That means fewer airfoils, fewer life-limited parts and, ultimately, lower maintenance costs. The real-world performance results are also impressive:

- Over 15 percent improvement in fuel burn
- Up to 75 percent reduction in noise footprint
- Annual per-plane reduction in carbon emissions of over 3,000 metric tonnes

Already five major aircraft manufacturers have placed orders for the game-changing PurePower engine. Mass production is slated to begin later this year.

▲ **PurePower engine** COURTESY PRATT & WHITNEY.

What advice would you give other engineering teams that want to increase their organization's focus on robust design?
I'm an advocate of what I call "design simulations": putting the right tools in the hands of designers to speed up the overall product development process. If your organization is serious about robust design, the first step is to make sure you have the right technology tools in place to manage large parametric simulations and drive rapid results.

Because robust design considers so many variables, any organization focused on this area is going to be running large simulations. An investment in high-performance computing resources is essential so that work can be accomplished and shared quickly. In just the last four years, Pratt & Whitney has quadrupled its computing capacity for a simple reason: We did not want computing power to be an obstacle to innovation and product integrity. For a small investment relative to the impact on our products, we are running large multiphysics simulations that support our DFV initiative — which allows us to reduce the risk of design mistakes that could result in large downstream warranty costs.

While technology is important, education and training are also critical. I believe that the engineering community needs to place a greater emphasis on statistical analysis, which lies at the heart of robust design. Our engineering students today are not being adequately trained in this area, and I'd like to see that change. As performance demands in every industry become more complex — and cost pressures escalate — engineers need to become proficient at quantifying the impacts of different materials, part geometries and other factors on ultimate performance. They also need to understand and analyze for the interactions of multiple physical effects, since the systems we are developing are becoming increasingly complex.

Finally, at the organizational level, a key robust design concept is standardizing work processes, which has been a real focus at Pratt & Whitney for the last decade. When you are exchanging

Robust Design at Pratt & Whitney

While robust design is an emerging concept for most companies, Pratt & Whitney began to focus on this idea as early as 1996. That year, as part of the company's internal quality program, every engineer was required to undergo training in statistical topics such as confidence intervals, probability distributions and regression modeling — and to understand how these concepts could help them solve common problems. Today, the company's Design for Variation effort has grown into a core competency, applied as a 10-step process that guides all engineering activities at Pratt & Whitney.

The company estimates that its component-level DFV initiatives have yielded a 64 percent to 88 percent return on investment by reducing design iterations, improving manufacturability, increasing reliability, improving on-time deliveries, and providing other performance benefits. As Pratt & Whitney focuses increasingly on the systems level, it estimates that it will realize a 40-times return on investment by achieving systems-level reliability goals much earlier in the development cycle. An ultimate benefit is shortening the overall development cycle.

IDENTIFYING CRITICAL CONDITIONS THAT LIMIT PART LIFE

Many components of jet engines require cast materials with long lead times. This results in the need to design parts and commit to geometry long before thermal boundary conditions are measured, so these designs need to be robust across a range of potential thermal conditions.

The Mid-Turbine Frame (MTF), a component of Pratt & Whitney's revolutionary PurePower® engine located between the high- and low-pressure turbines, provides a fairing around the structural frame and bearing oil tubes. This frame carries the pressure loads on the part created by turning the air; however, the majority of these loads are driven by transient thermal gradients as the part heats from idle to takeoff conditions and then cools again. The design requires various areas of the MTF to grow and shrink, as well as to smoothly distribute any thermal load generated so that stresses do not concentrate.

Life expectancy of an MTF airfoil is determined, in part, by the shape of its thermal profile, the magnitudes of local mechanical stresses and the inherent material capability. These are, in turn, determined by a number of factors: part-to-part variation (airfoil geometric variation within tolerances), engine-to-engine variation (thermal profiles), inherent material capability variation, and uncertainty in the lifing models. The combination of these types of uncertainty can cause wide variation in airfoil life.

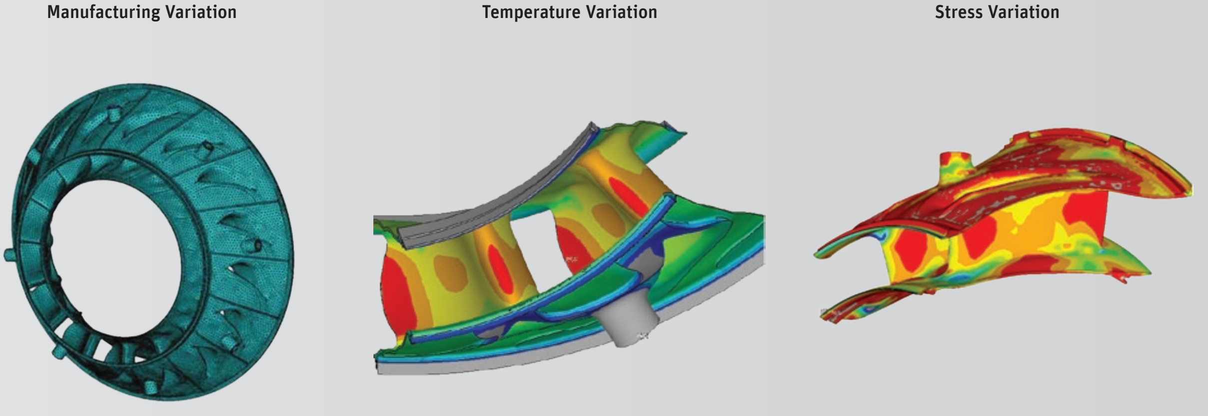
In designing the MTF, Pratt & Whitney's goal was to find the nominal set of MTF features that would meet part life, weight and efficiency objectives while being robust with regard to all important sources of variability and uncertainty. The strategy was to make all models parametric, combine them into a single automated workflow, run a designed experiment over the model input space using the automated workflow with high-performance computing, and use the results to guide Pratt & Whitney engineers to a feasible/optimal region of the design space.

The parametric models included an NX® geometry model with automated meshing and ANSYS thermal and structural finite element models. A unique system called CCE (Collaborative Computing Environment) created a linked, distributed, automated workflow. All the building and execution of the analytical models resided with their owners and were linked together by scripts in a revision management application. The model input space covered the geometric design space, as well as the ranges of all variable features and uncertain parameters and boundary conditions.

The use of automated workflows with relatively large, multidisciplinary design spaces — as in the development of the MTF — requires efficient tools and techniques for solution visualization work with engineers around the world, you need fast, reliable software tools as well as highly defined workflows and processes. We have created hundreds of internal courses in which we teach standard processes and methods that reinforce our commitment to quality and consistency.

How would you describe your relationship with ANSYS?
Since we are an advanced user of engineering simulation, we have collaborated with ANSYS on many projects and have given ANSYS a lot of product feedback. ANSYS software is a widely used commercial tool, which has led to a much broader implementation of DFV at Pratt & Whitney. Our younger engineers are familiar with ANSYS solutions, and they can easily fit the tools into our standard workflows. They like being exposed to multiple physics and seeing all the parts of a specific problem.

Probably the most important contribution that ANSYS has made is allowing Pratt & Whitney engineers to push the envelope of previous engine designs, all in a very-low-risk virtual environment. We can see quickly what is possible, without making a huge investment in prototype construction and testing. Recently, we used multiphysics simulations — combining ANSYS Mechanical and ANSYS Fluent for example — to convince a major customer that they were making a design request that was not practical, because their modification would add significant weight to the engine. By showing them the real-world effects of their request via ANSYS simulations, which the customer also used, we avoided increasing complexity that we believe would have led to numerous issues. Without ANSYS software, some of these issues would not have been visible until installation. Today we are using ANSYS solutions in ways we never thought possible. ANSYS is definitely supporting our efforts to stay out in front of our industry as a leader and innovator.



▲ In a search for a nominal design that is robust to variability and uncertainty, Pratt & Whitney created an automated workflow for its Mid-Turbine Frame that would ensure design robustness by considering a range of manufacturing, temperature and stress variations.

and evaluation. The goal is identifying the life-limiting locations and, at a more detailed level, detecting the parameters that are driving the part's life.

One of the tools for critical driver identification and insight into interaction between parameters is global variance-based sensitivity analysis. Global sensitivity analysis uses the results generated by executing the analysis workflow over a prescribed designed experiment. These same results are used to develop emulators for identification of feasible design regions. When needed, more detailed exploration can be executed for refinement of local design solutions.

The collection of automated workflows, variability and uncertainty analysis, and emulators allowed the Pratt & Whitney team to address its design challenges more quickly than by using traditional analysis strategies. For example, when aerodynamics refinements led to topological changes, the team used the established tools and process to efficiently adapt the toolset and continue the design activities. This enabled the team to design an A320 MTF that is robust with regard to uncertainty in thermal profiles — while exceeding life, weight and efficiency requirements and adhering to the design schedule.

BREAKING THE CODE

Turbomeca reduces development time by using ANSYS SCADE Suite for helicopter engine control software development.

By Didier Bernard, Head of Software System Group, Turbomeca, Bordes, France

The engine control system is a critical part of today's helicopter engines that controls fuel injection and other engine functions. In developing the embedded software that runs the engine control system, Turbomeca switched from manual coding to a model-based design approach that involves the creation of an executable model in a block diagram design environment. Engineers define the functionality of the control system within ANSYS SCADE Suite using blocks that represent algorithms or subsystems. They validate the model and use it to automatically generate embedded code. Turbomeca engineers have developed control systems that are powering the company's two latest engines using the new method. The new process has demonstrated the ability to substantially reduce errors and development time.

Turbomeca, part of the Safran group, is the world's leading producer of helicopter engines and has produced 70,000 engines since its founding in 1938. The company specializes in the design, production, sale and support of low- to medium-power gas turbines to power helicopters. Turbomeca turbines propel civil, parapublic (such as police and fire department) and defense helicopters for all the leading helicopter manufacturers. Including its joint programs with other manufacturers, 18,200 of its turbines are now in operation, and its engines have provided 90 million operating hours.

DIGITAL ENGINE CONTROL UNIT

Turbomeca's engines are organized into engine families based on the level of power output. Each family of engines

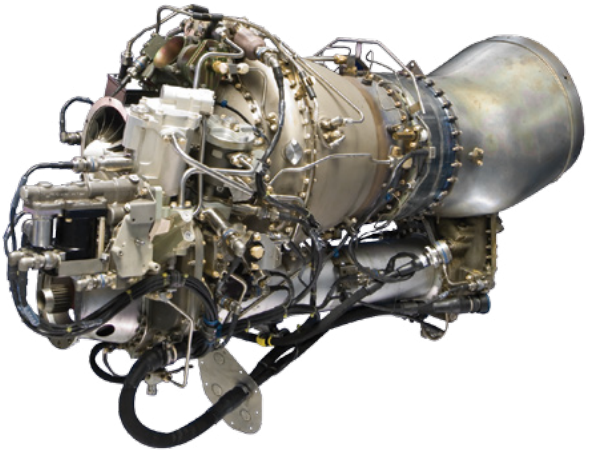
Turbomeca switched from manual coding to a model-based design approach.

Development time was reduced by 30 percent.

includes several different variants that meet the specific requirements of different types of helicopters. The engines have a modular architecture with the key modules being the compressor, combustion chamber and turbine. The digital engine control unit, called the full-authority digital engine control (FADEC), regulates the engine's speed by modulating fuel flow based on the environment, torque evolution and use case.

The FADEC includes two identical control channels, each of which is capable of independently controlling the engine. The FADEC can transfer control from one channel to the other if a channel is not functioning correctly. Each FADEC platform consists of hardware and an operating system, and it is specified to be compliant with several engine families. Application software is developed for each engine variant to take into account helicopter characteristics and customer needs. There are many commonalities between engine control units of different variants, and a modular architecture has been defined with re-usable components to encourage re-use between different variants.

Full-authority digital engine control unit



Arriel 2D engine

SOFTWARE DEVELOPMENT PROCESSES

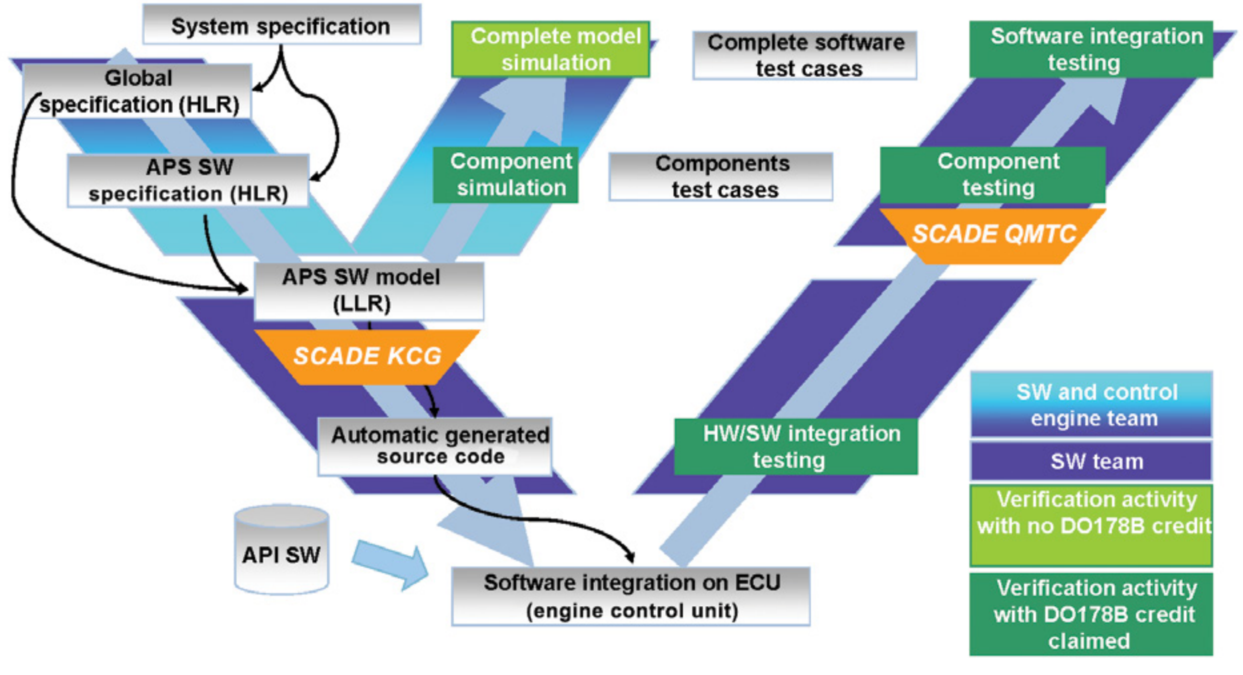
Since 1985, Turbomeca has progressed through four distinct software development processes for developing application software for its FADECs to address improved technological solutions, evolution in airworthiness requirements, and increased software functionality.

In 2005, Turbomeca developed the G4 software process by implementing a development environment that includes requirements management, model-based design, simulation, automation of tests, and qualified code generation in compliance with DO-178B (the standard used to qualify all avionics software by the FAA, EASA and other certification authorities). SCADE Suite is used in the new software process because model-based development is clearer and more understandable than working with source code for systems teams and promotes co-engineering between systems and software teams.

Model simulation provides an efficient method of detecting functional faults at the earliest possible moment. SCADE Suite delivers an efficient model checker that enables engineers to detect problems early in the design phase rather than later in the integration phase. Test cases can be run in the PC environment rather than in the much more expensive and complicated target hardware environment that is deployed on the aircraft. SCADE Suite incorporates a reusable symbol library that promotes re-use and commonality of design within and across software projects.

SCADE Suite helps reduce development duration and cost by enabling efficient codesign engineering between control law and software teams through the use of formal language and methods that are clearer and more understandable than source code and by reducing the code integration phase. Defects are checked at the design level so that they are detected at the earliest possible stage. Using a qualified code generator, SCADE KCG guarantees compliance between the design model and the code, and strongly reduces formal verification at code level. Consistency of modules integration is verified at the model level before generating the C code, eliminating the need for integration verification at the code level. The code generator is qualified as a DO-178B development tool (cf. section 12.2 of DO-178B), so the conformance of the code to the input model is trusted, eliminating the need for verification activities related to the coding phase.

The development team achieved a decrease of 50 percent in the number of open problems on certified versions.



▲ G4 process overview

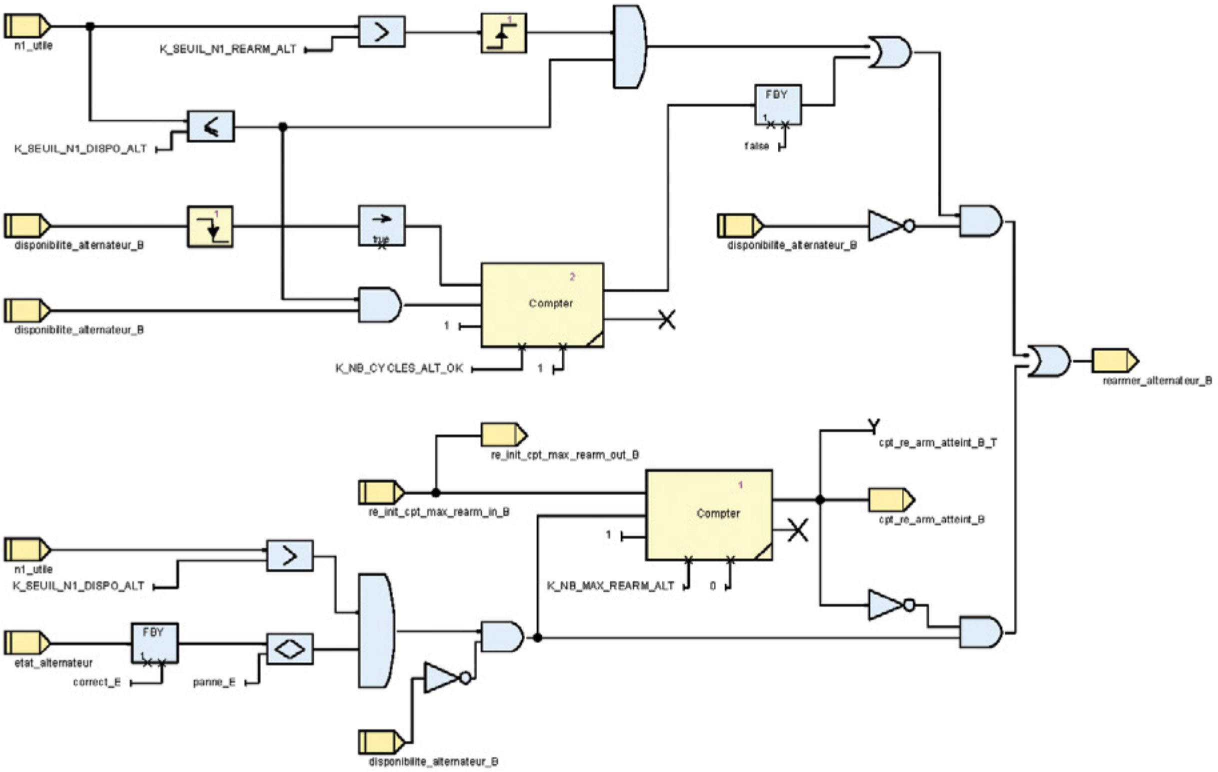
CODE SHARING AND RE-USE

The G4 process incorporates a generic modular software architecture based on configurable functions that can be easily re-used for multiple software development projects. It has enabled teams to focus on activities for which they have specific skills and easily share data with other teams working on other parts of the project. The application software developed by the G4 process is independent from the target hardware platform, which reduces the time and expense associated with changing hardware platforms.

The G4 process was first used in the development of the FADEC for the Arriel 2D engine, which is designed for light, single-engine helicopters in the 5,000-pound weight class and currently powers Eurocopter AS350 B3e and EC130T2 helicopters. The Arriel family of engines is used on more than 60 percent of all helicopters in the world in the 700-to-900 shaft horsepower (shp) power class. The dual-channel FADEC provides engine state monitoring, including fuel and oil filter clogging. It also regulates gas generator speed and power turbine inlet temperature for better power optimization and increases mean time between unscheduled removal. The time between overhaul at initial entry into service is 4,000 hours, but the target for engines in service is 6,000 hours.

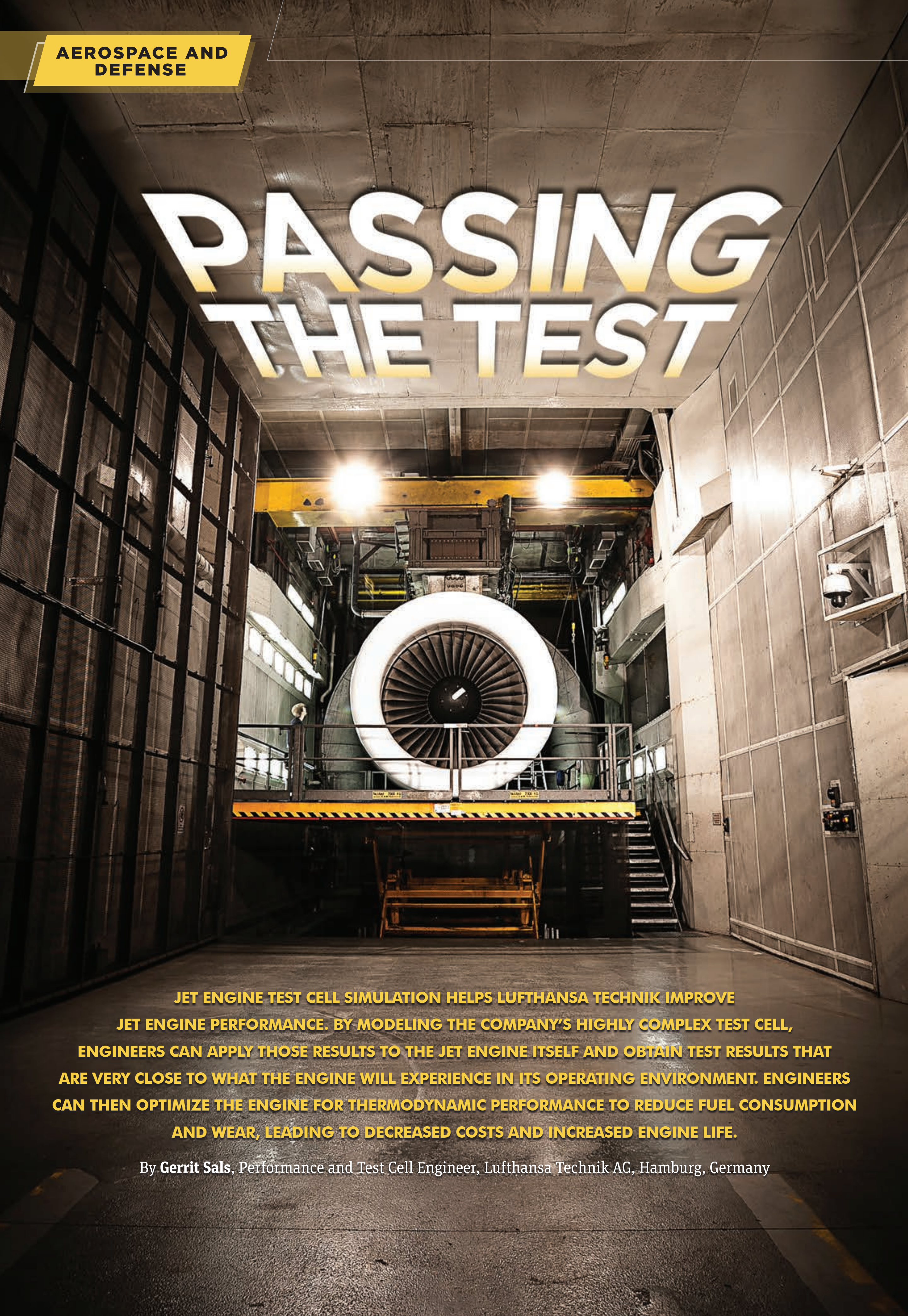
Turbomeca engineers first used the G4 process in developing embedded software for the Arriel 2D engine that was certified in 2011 and the subsequent Arriel 2E that was certified in 2012. The development team achieved a decrease of 50 percent in the number of open problems on certified versions by detecting problems earlier in the development process and correcting them prior to certification. Development time also was reduced by 30 percent by taking advantage of G4 process improvements, including SCADE Suite. FADEC software for five more engines is currently under development at Turbomeca using the G4 process. The company is also evaluating the latest version of SCADE Suite for future projects because of its potential to bring improvements in compute time and language capability.

REDUCING PRODUCT DEVELOPMENT RISK AND COMPLEXITY WITH MODEL-BASED SYSTEMS ENGINEERING AND EMBEDDED SYSTEMS



▲ **SCADE model for control function that determines whether electricity produced by engine is used to power accessories**

AEROSPACE AND DEFENSE

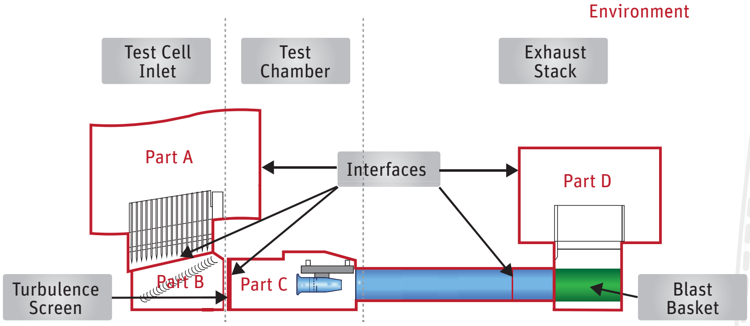


PASSING THE TEST

JET ENGINE TEST CELL SIMULATION HELPS LUFTHANSA TECHNIK IMPROVE JET ENGINE PERFORMANCE. BY MODELING THE COMPANY'S HIGHLY COMPLEX TEST CELL, ENGINEERS CAN APPLY THOSE RESULTS TO THE JET ENGINE ITSELF AND OBTAIN TEST RESULTS THAT ARE VERY CLOSE TO WHAT THE ENGINE WILL EXPERIENCE IN ITS OPERATING ENVIRONMENT. ENGINEERS CAN THEN OPTIMIZE THE ENGINE FOR THERMODYNAMIC PERFORMANCE TO REDUCE FUEL CONSUMPTION AND WEAR, LEADING TO DECREASED COSTS AND INCREASED ENGINE LIFE.

By **Gerrit Sals**, Performance and Test Cell Engineer, Lufthansa Technik AG, Hamburg, Germany

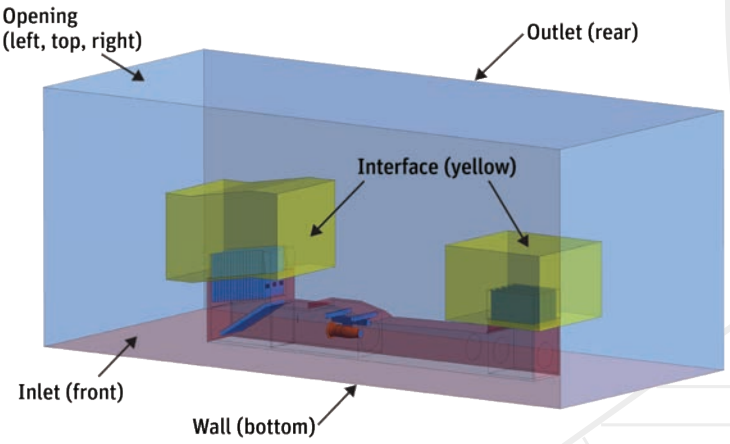
Overhauling a typical commercial jet aircraft engine might cost about $2 million as an expert team inspects and services or replaces up to 40,000 parts. Such an overhaul could be necessary each time the engine flies between 2,000 and 10,000 flights. Overhauls can vary greatly in their work scope, which describes the engine components that are to be serviced or replaced. The work scope is vital because it largely determines the overhaul cost and the performance of the overhauled engine. Lufthansa Technik is improving the engine overhaul process by simulating individual engines at a very detailed level to quantify the relationship between the condition of specific components and the operating behavior of the engine. The insight gained from these simulations allows the team to develop a customized work scope in close consultation with the customer. This work scope might allow engineers to increase the thermodynamic engine performance, which reduces fuel consumption and wear, thereby decreasing future maintenance costs. The understanding acquired from simulation also makes it possible to obtain maximum use from thermo-dynamically as well as economically critical parts, for example, by operating expensive turbine blades for longer periods.



^ The test cell was partitioned into five models joined with interfaces to enable simulation of the complex model.

Until recently, these simulations were based solely on the engine operating in the air or on the runway, in contrast to jet engine diagnosis and acceptance testing, which is performed in test cells where operating conditions can be significantly different. Lufthansa Technik engineers have long wanted to simulate engines as if they were operating on the company's jet engine test cell. This would require modeling the test cell so the results could be used in modeling the engine. However, test cells are challenging to simulate due to the size and complexity of the geometry, the large range of length and velocity scales present, and flow Mach numbers ranging from near zero to transonic.

Lufthansa Technik engineers have recently overcome these challenges by simulating one of the company's test cells and validating the results against physical testing measurements. Once the team is able to use the test cell simulation results as input to the engine simulation, engineers will be able to better understand the results of diagnostic testing in the test cells, and will also be better able to predict the effects of different overhaul work procedures on acceptance testing. The result should be improvements in engine performance and more accurate overhaul work scoping with resulting cost reductions.



^ Outer boundary conditions

OPTIMIZING THE OVERHAUL PROCESS

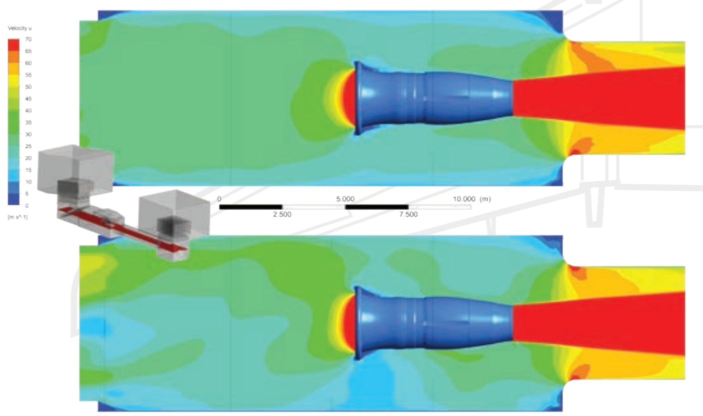
Lufthansa Technik AG is one of the world's leading providers of aircraft maintenance, repair and overhaul services. To improve engine efficiency while avoiding unnecessary work during engine overhauls, detailed knowledge of the internal interactions in the engine is essential. Lufthansa Technik constantly monitors important components so they can be replaced as a function of their condition. Further efficiency improvement can be achieved by precisely determining how the condition of individual components will affect the engine behavior as a whole. By establishing this link between component condition and the operating behavior of the engine, it is possible to target critical components to address during overhaul.

Lufthansa Technik engineers perform three levels of simulation to determine a cause-and-effect link between component condition and engine operating behavior. The highest level is the overall engine level, in which general engine parameters such as thrust, fuel consumption and exhaust gas temperature (EGT) are determined using commercially available thermodynamic cycle analysis software. The second level is a flow simulation of the entire engine based on the multiple mean-line approach. The third level consists of detailed ANSYS CFX computational fluid dynamics (CFD) simulations of sections of the engine.

"The understanding acquired from *simulation* makes it possible to obtain *maximum engine* life."

Recently, Lufthansa Technik engineers set out to further improve this process by simulating the company's test rig to obtain boundary conditions for engine simulations. Internal boundary conditions are derived from the cycle analysis in 95 percent of the cases, which in turn is based on test-cell data. Employing data obtained from a 3-D flow field of the test cell helps the engineers simulate behavior under specific conditions, such as considering the inlet flow of the fan to determine the effects of humidity, rain and crosswinds. This, in turn, enables them to better predict the relationship between component condition and performance on the test cell. Because of the complexity of the test cell geometry, it was split into five models with interfaces between them so the adjoining models provide boundary conditions for each other.

By partitioning the test cell, engineers reduced the model complexity and size, and enabled a modular approach whereby different simulation configurations can easily be constructed by assembling individual components. The CFX flexible general grid interface (GGI) enables such a modular approach. Part A contains the inlet to the test cell and inlet splitters; Part B includes turning vanes; Part C comprises the test chamber, turbulence screen, thrust stand, engine and augmenter tube; and Part D contains the exhaust stack and outlet splitters. The area surrounding the test stands was modeled separately and called the Environment. In addition, the turbulence screen and blast basket were each incorporated into the simulation as subdomains.



^ Axial velocity inside test chamber for static conditions (top) and crosswind (bottom)

MODELING THE TEST CELL

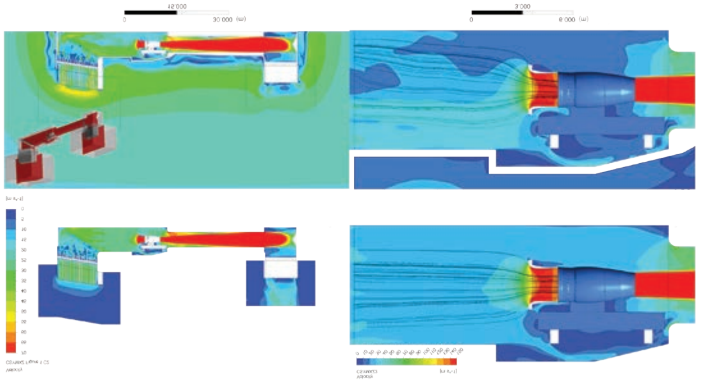
Engineers generated each mesh segment individually using ANSYS ICEM CFD Hexa capabilities, part of ANSYS meshing. Creating the mesh was the biggest challenge in this simulation process. Lufthansa Technik engineers used the mesh diagnostic and repair tools to maintain high levels of mesh quality throughout the mesh generation process. The mesh structure for Parts A, B, D and the Environment was generated as hexahedral H-grids because a hex mesh provides the best trade-off between accuracy and resource requirements. Additionally, small changes can be performed easily. On the other hand, Part C was meshed as a structured hexahedral O-grid for maximum accuracy in this critical section of the model. The interfaces reduced computational time by making it unnecessary to propagate the structured hexahedral O-grid through the turning vane geometry in Part B.

The air enters the test cell through the inlet, where it accelerates when passing through the flow splitters. The turning vanes deflect the vertical flow without significant acceleration. Downstream, the flow passes through the turbulence screen, which leads to a drop in total pressure along with more uniform air flow. The engine then adds energy to the air flow,

Testing the Next Generation of Rockets
ansys.com/testing

increasing the temperature, velocity and total pressure behind the engine. This in turn leads to an acceleration of the air bypassing the engine, which is called the ejector effect. The exhaust gas then leaves the test cell through the augmenter tube, blast basket and exhaust stack.

Engineers simulated the test under two different sets of environmental conditions, which were used as boundary conditions. The first assumed no air movement at the inlet and outlet of the test cell, and the second assumed a 20 m/s crosswind at the inlet and outlet. While different wind directions and speeds are not used in testing, adjustments were made to the CFD model to account for crosswinds, and simulation was used to evaluate those adjustments. The external boundary conditions, which are needed only during the crosswind simulation, include an inlet in front, an outlet at the rear, and openings in the left, top and right of the model. The model's internal outlet boundary (engine inlet) is dependent on the model's internal inlet boundary (engine outlet). The mass flow of these boundaries is coupled through functions based on the static pressure and total temperature at the engine's exhaust nozzle. The functions were derived using thermodynamic cycle analysis. This setup increases the accuracy of the model as the engine changes its operating point according to the test cell flow conditions.



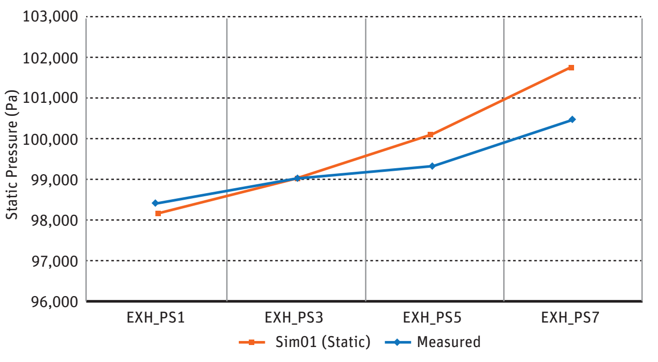
^ Fluid flow in the test cell predicted by simulation for static conditions (top) and crosswind (bottom). This enables engineers to better understand the test cell under real-life conditions to aid jet engine overhaul.

VALIDATING THE SIMULATION

To better understand the test cell results, all that is needed from the test cell simulation is to determine the boundary conditions at the engine inlet and outlet. However, Lufthansa Technik engineers wanted to validate the complete model — including its ability to predict pressures and velocities at any point in the solution domain — so that this information could also be used in evaluating proposed changes to the test cell. The test cell model was validated by comparing simulation results and test cell measurements of static pressure at various points inside the augmenter tube. The deviation between the simulation and test results was very good (from –0.05 percent to –1.33 percent at four different points). However, Lufthansa Technik engineers are working on further improvements in accuracy by refining the mesh in the area of the blast basket and further downstream.

The test cell model will soon be used to provide boundary conditions for engine simulations used as part of the work scoping process for engine overhauls. Accurate engine-in-test-cell simulation will help engineers further improve the performance of overhauled engines and refine the work scoping process with the potential for significant cost savings. For example, the customer may specify that the overhauled engine must provide a certain EGT on the test cell. Lufthansa Technik engineers will be able to better evaluate the impact of different possible work scopes on the EGT as measured on the test stand. In addition, the test cell model will be used to improve the test cell design and evaluate the impact of different sensor placements in specific tests.

Using simulation, Lufthansa Technik will not only improve jet engine performance for customers but fine-tune internal processes to reduce costs. Simulation accuracy reduces risk and makes the company more competitive.



^ Comparison of simulated and measured pressure inside the augmenter tube shows acceptable agreement.

Efficient and High-Performance Flow Path Development
ansys.com/flowpath

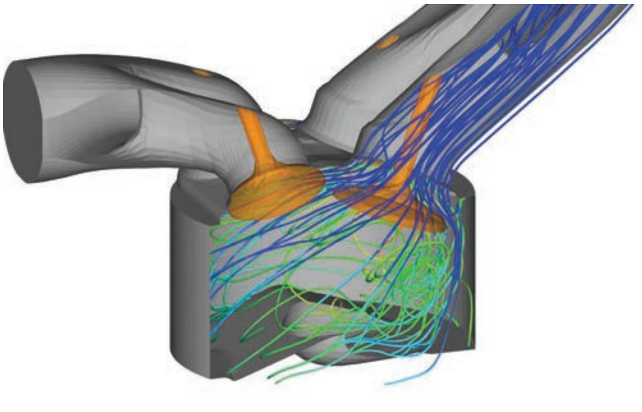
FAST, ACCURATE SIMULATIONS FOR FUEL COMBUSTION APPLICATIONS

The acquisition of Reaction Design broadens the ANSYS simulation offering with industry-leading chemistry solvers to advance clean engine and fuel technologies.

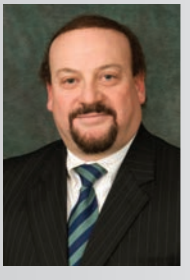
By Bernie Rosenthal, CEO, Reaction Design

Now — more than ever — automobile engine and turbine manufacturers are under intense pressure to develop and deliver higher-performance products with significantly reduced emissions.

In early March 2014, the U.S. Environmental Protection Agency (EPA) announced new, tighter fuel standards for motor vehicles as part of ongoing initiatives to lower greenhouse gas emissions. According to the Union of Concerned Scientists, today's on-road vehicles produce over a third of the carbon monoxide and nitrogen oxides in our atmosphere along with over 20 percent of global-warming pollution [1], while power generation is the single largest source of U.S. global warming emissions [2].



▲ FORTÉ CFD package for advanced, 3-D internal combustion engine design based on detailed fuel chemistry mechanisms



ANSYS is the global leader and innovator of engineering simulation software. Reaction Design creates solutions that automate the analysis of chemical processes through computer simulation and modeling. By joining forces, we provide our customers with the most powerful and effective combustion simulation tools available in the world.

Merging the technical strengths of Reaction Design and ANSYS into a single company creates new opportunities to enable the development of less-polluting, higher-efficiency and more-competitive products in transportation, energy and materials processing sectors.

Effective simulation of underlying detailed chemistry is critical to advancements in engine and fuel technology, and ANSYS customers now have easy access to kinetics tools and fuel libraries from Reaction Design. Understanding and predicting the effects of fuel chemistry in a combustion system with fast, accurate, cost-effective modeling is vital to developing competitive products that translate reliably to real-world functionality.

As the two companies' technologies come together, exciting new integrated capabilities will become available to ANSYS customers, helping to drive increased fuel efficiency around the globe and leading to new advancements in engine and fuel technology.

Bernie Rosenthal, CEO, Reaction Design

Model Fuel Library

Fuel Chemistry Component Class	Number of Components	Relevant for Modeling: Gasoline	Diesel	Jet Fuels or FT Fuels	Natural or Synthetic Gas	Biofuels or Additives
Hydrogen/CO	2				●	
n-Alkanes	9	●	●	●	●	
iso-Alkanes	3	●	●	●	●	
1-Ring Aromatics	5	●	●	●		
2-Ring Aromatics	2	●	●	●		
cyclo-Alkanes	3	●	●	●		
Olefins	6	●	●	●		
Oxygenated Fuels	8				●	●
Soot Precursors and Emissions Pathways	10	●	●	●	●	●

▲ The Model Fuel Library features more than 40 validated fuel component models.

Designing high-performance internal combustion engines and gas turbines that meet regulatory mandates for reduced levels of greenhouse gases and other toxic emissions is perhaps the top challenge that transportation manufacturers and energy producers face today.

Reaction Design products enable designers to achieve their clean technology goals by automating the analysis of chemical processes that take place in a wide range of products and applications. The company serves more than 400 customers from around the globe, including industry-leading internal combustion engine, industrial, and aviation turbine manufacturers, materials processors and energy producers.

Combustion CFD simulation makes it possible for design engineers to create lower-emission combustion systems without spending millions of dollars on physical mockups and costly trial-and-error testing. However, ensuring that simulations accurately predict real-life fuel effects demands using complex algorithms that describe the physics and thermodynamic behavior of combustion. It also requires a detailed understanding of the chemical makeup of the fuels to be burned and types of engine to be deployed.

FAST, HIGHLY ACCURATE SIMULATIONS

Reaction Design products are designed to maximize simulation accuracy and reduce the overall time required to create a fully actionable design. Combustion simulation is a valuable aid to designers in meeting their goals, but only if the results of their modeling gives true insight into the engine's behavior. Obtaining accurate results from combustion simulation requires capturing both the physical and the chemical characteristics that can change radically over a full engine-duty cycle. In an internal combustion engine, for example, spray breakup and evaporation, turbulence, ignition delay and flame propagation are all factors that must be modeled accurately to yield meaningful results.

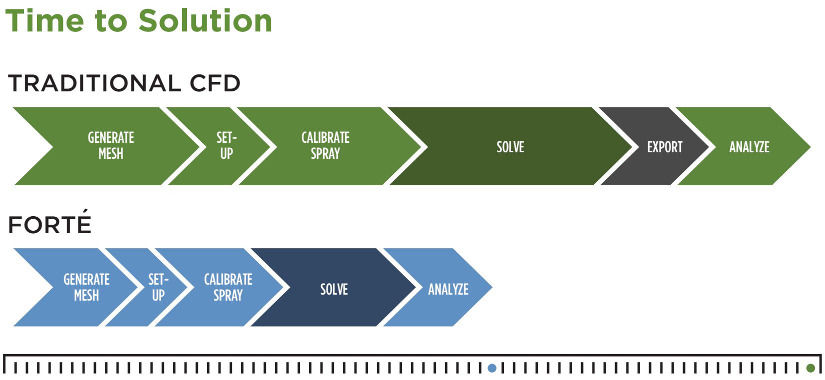
Thanks to massively parallel computers, engine geometries can be represented with amazing detail using computational meshes in CFD that approach 100 million cells. However, the chemistry solver technology included with most CFD packages is slow relative to the flow calculations. So it is common for engineers to use single-component, severely reduced fuel models in their combustion simulations. These reduced fuel models lack the detail required to accurately predict key engine performance factors, such as ignition delay, flame propagation, NOx, CO and PM (soot) emissions.

In 2005, Reaction Design launched the Model Fuels Consortium, a 20-member group that includes global leaders in energy and engine manufacturing: Chevron, ConocoPhillips, Cummins, Dow Chemical Company, Ford Motor Company, GE Energy, General Motors, Honda, l'Institut Français du Pétrole (IFP), Mazda, Mitsubishi Motors, Nissan, Oak Ridge National Laboratory, Petrobras, PSA Peugeot Citroën, Saudi Aramco, Suzuki, Toyota and Volkswagen. The consortium's goal was to enable more timely and cost-effective design of cleaner-burning, more-efficient engines and fuels through use of chemically accurate fuel component models in software simulation and modeling.

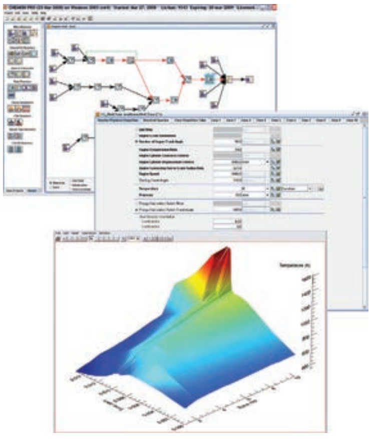
Reaction Design worked with data from the Model Fuels Consortium to develop its Model Fuel Library (MFL), a compendium of detailed chemical kinetics and mechanisms for 56 fully validated, self-consistent fuel components derived from a master reaction mechanism roster of more than 4,000 chemical species. The MFL enables engine designers to accurately simulate fuel effects in virtually all types of automotive, aircraft and power-generation engines; the components can be combined to model a large variety of new or existing fuel blends.

When coupled with Reaction Design's software suite, the fuel model

Combustion CFD simulation makes it possible for design engineers to create lower-emission combustion systems without spending millions of dollars on physical mockups.



▲ FORTÉ delivers dramatic reductions in time to solution over conventional CFD approaches.



▲ CHEMKIN is the gold standard simulation software for complex chemical processes.

components greatly increase the accuracy of results across a wide range of operating conditions and fuel types without negatively impacting time to solution, measured as the total wall clock time an engineer experiences from simulation setup through completion of analysis of the visualized results. The Reaction Design products address traditional bottlenecks in the simulation process by offering easy-to-understand, wizard-like graphical user interfaces to ease the setup, solve and analyze steps of the simulation process, and incorporate mathematical solver technologies with the appropriate level of physics and chemistry detail to ensure accuracy.

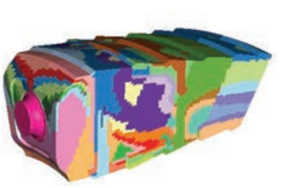
INDUSTRY-LEADING PRODUCTS

CHEMKIN, the gold standard for modeling gas-phase and surface chemistry, was born at the Sandia National Laboratories in the 1980s as a set of command-line–driven codes that describe the complex series of molecular-level chemical reactions that take place during fuel combustion. These led to the development of a suite of detailed-kinetics reactor models that use idealized representations of reacting flows. In time, the 0-D and 1-D flow approximations became the workhorse of fundamental combustion research. CHEMKIN also became an important educational tool in chemical engineering, mechanical engineering and chemistry curricula. The detailed approach to gas-surface reactions led to wide use of CHEMKIN for materials processing studies, such as chemical vapor deposition or plasma etching in microelectronics chip manufacturing.

In 1997, Reaction Design became the exclusive developer and licensor of CHEMKIN technology. CHEMKIN evolved into commercial-quality software that enables engineers and scientists to develop a comprehensive understanding of chemical processes and kinetics and to quickly explore the effects of design variables on performance, by-products production, and engine or process efficiency. Using CHEMKIN, researchers are able to consider thousands of chemical species — and tens of thousands of reactions — for wide ranges of processes and conditions. The advanced solvers developed for CHEMKIN enable fast, accurate simulation of underlying detailed chemical and ignition behaviors, cutting combustion simulation times from days to hours — or hours to minutes — for complex models with large mechanisms. The speed and accuracy of these simulations allow designers to test alternative system configurations and inputs to optimize for performance, efficiency and emissions compliance — virtually, before moving to the prototype stage of their development program.

Reaction Design's ENERGICO simulation package brings the power of detailed kinetic modeling to combustion system design for applications such as gas turbine combustors, burners for boilers and furnaces, and flares and incinerators. ENERGICO uses CFD simulation to help create accurate chemistry models of a system to meet the challenges of emissions reduction and combustion stability for energy production and related industries. Using its strong background and experience in chemistry, Reaction Design created the FORTÉ CFD Package that makes possible realistic 3-D modeling of fuel effects in internal engines. FORTÉ uses proven mathematical techniques and algorithms, coupled with detailed chemical kinetics to simulate the combustion process that takes place inside an engine chamber and predict the effects of operating conditions and fuel variations on the engine's behavior. FORTÉ's superior time-to-solution metrics make it a trusted part of the internal combustion engine design workflow and an invaluable aid in producing cleaner-burning, higher-performance and more-efficient engines.

The Reaction Design team is proud to provide solutions that have helped leading companies to create better products by automating the analysis of chemical processes using computer simulation and modeling. And now, as a part of ANSYS, the team is expanding its vision to provide capabilities to an even broader audience.

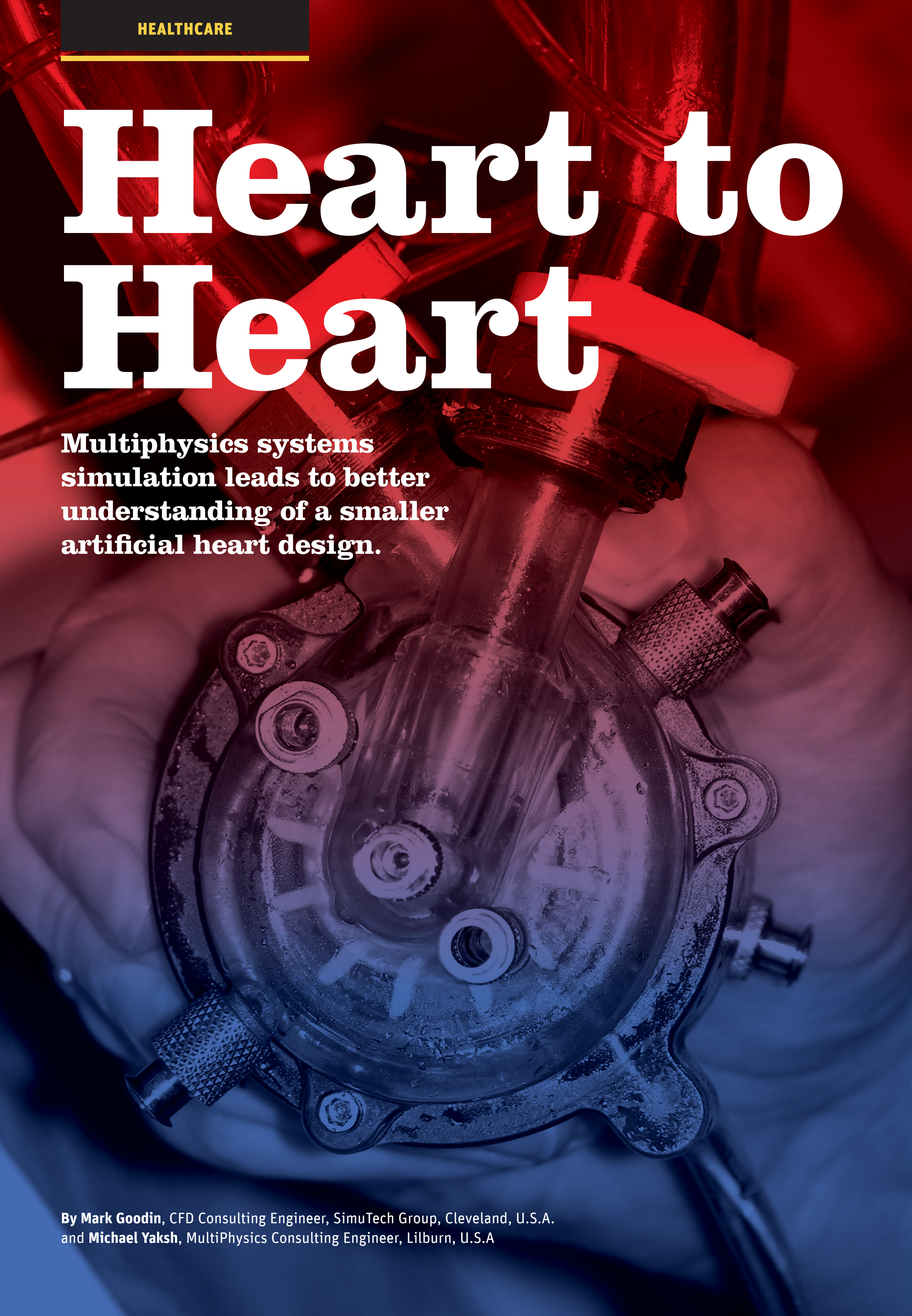


▲ The ENERGICO simulation package chemically simulates combustion in a virtual environment for multiple fuels.

REACTING FLOWS AND COMBUSTION

References

[1] Union of Concerned Scientists: Cars, Trucks, and Air Pollution, www.ucsusa.org/clean_vehicles/why-clean-cars/air-pollution-and-health/cars-trucks-air-pollution.html
[2] Union of Concerned Scientists: Clean Energy, www.ucsusa.org/clean_energy/



Heart to Heart

Multiphysics systems simulation leads to better understanding of a smaller artificial heart design.

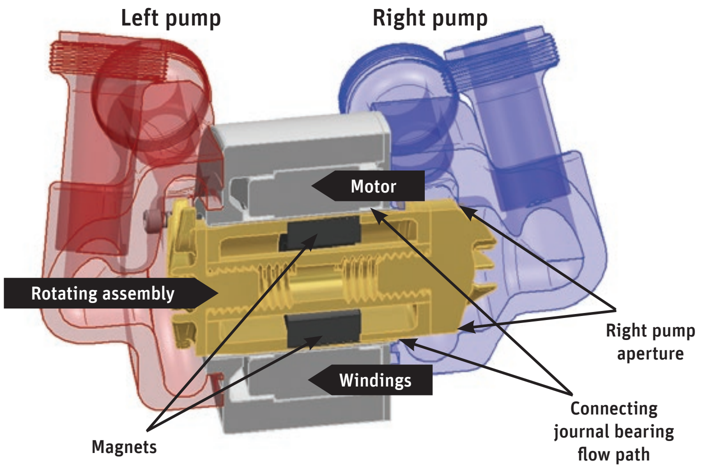
By Mark Goodin, CFD Consulting Engineer, SimuTech Group, Cleveland, U.S.A.
and **Michael Yaksh**, MultiPhysics Consulting Engineer, Lilburn, U.S.A

A new continuous-flow total artificial heart (CFTAH) is smaller and less complex than other artificial heart designs. It features a single moving part, the rotor, which is suspended by a combination of magnetic and fluid forces. This new heart is moving into animal testing, which is expensive and time-consuming, so the cost of failure at this stage is high. To minimize the risk and number of expensive design changes, engineering consultants from the SimuTech Group are performing multiphysics simulation that incorporates electromagnetic simulation coupled with fluid flow to fully explore the CFTAH's operation. To date, simulation has been used to calculate the pump's hydraulic performance, static pressures on pump surfaces, rotor torque, rotor axial forces, and other key parameters — all as part of the process of ensuring product design robustness before testing with live animals.

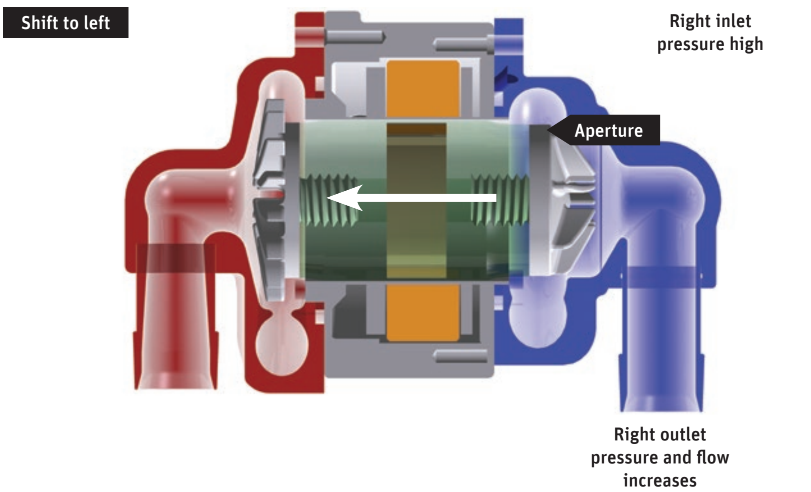
POTENTIALLY LIFE-SAVING DESIGN

More than 300,000 Americans die from heart failure each year, and of these, up to 20 percent die while waiting for a heart donor. Artificial hearts have the potential to save many of these people. But existing FDA-approved devices are complex, bulky and so large that they fit only 20 percent of women and 50 percent of men.

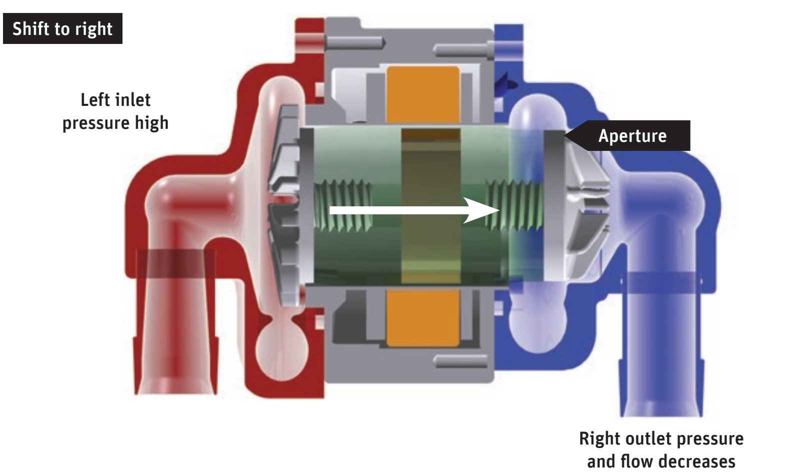
The continuous-flow total artificial heart's unique design features a single moving part: the rotor, which is suspended by a combination of magnetic and fluid forces.



▲ CFTAH geometry



▲ When right inlet pressure is high, the rotor moves to the left.



▲ When left inlet pressure is high, the rotor moves to the right.

A more-compact artificial heart design will fit adults and teenagers, and it is less complex than other designs.

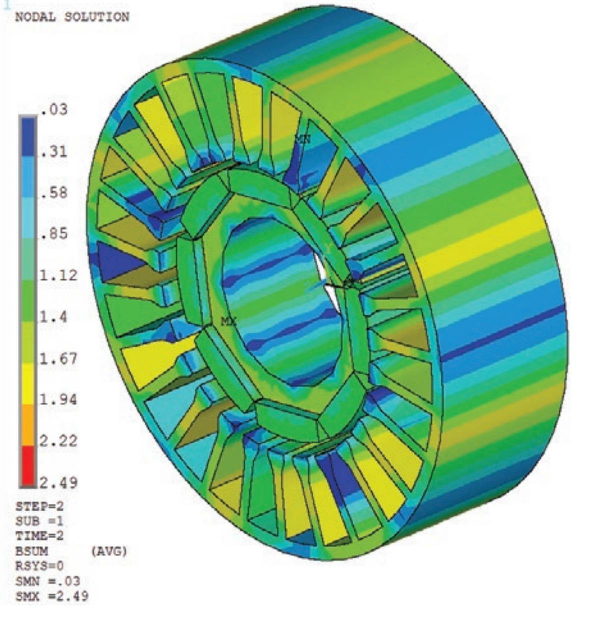
The CFTAH is a more compact and simpler artificial heart that fits most adults and many teenagers. The device is intended for use as a bridge to a transplant as well as for permanent use to completely replace a failing human heart.

The CFTAH's unique design delivers both simplicity and efficiency. A single motor and single power cable drive the organ's rotating pump assembly. Impellers supporting left and right circulation are mounted on opposing ends of the rotor. The rotor is radially suspended by a blood-lubricated hydrodynamic journal bearing designed to minimize blood shear while maintaining stable operation. During operation, the rotating assembly reaches a radial position in which the fluid-generated hydraulic-bearing forces are balanced by electromagnetic forces exerted by the pump motor.

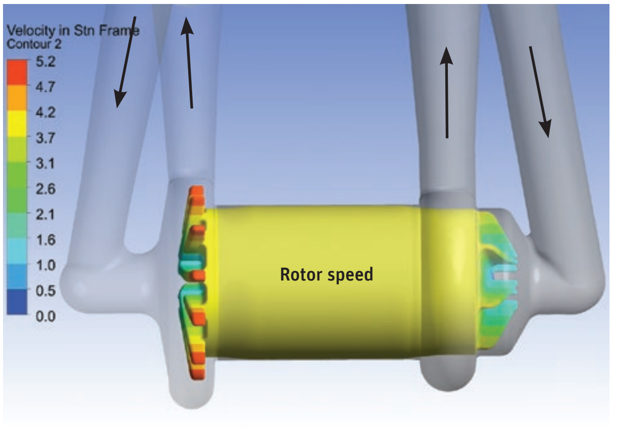
During normal operation, the rotor is free to move axially, and its axial position is determined by the magnet's axial restoring force and opposite-acting left and right pump side pressures. When the right pump pressure is higher than the left pump pressure, the rotating assembly is shifted by hydraulic forces to the left. This leftward shift increases the size of the right pump aperture, which increases the right pump's output pressure and flow rate. The increase in right pump performance raises the pressure and flow rate entering the left pump, which increases the left pump pressure and causes the rotating assembly to shift back rightwards. This self-regulation process automatically corrects any imbalances between the right and left side pumps. If there is a sudden change in pump pressure, the motor's axial restoring force limits the overall axial travel of the rotating assembly. This innovative design eliminates the need for components that have complicated other artificial heart designs, such as valves, sensors and actuation mechanisms.

ROLE OF ELECTROMAGNETICS SIMULATION

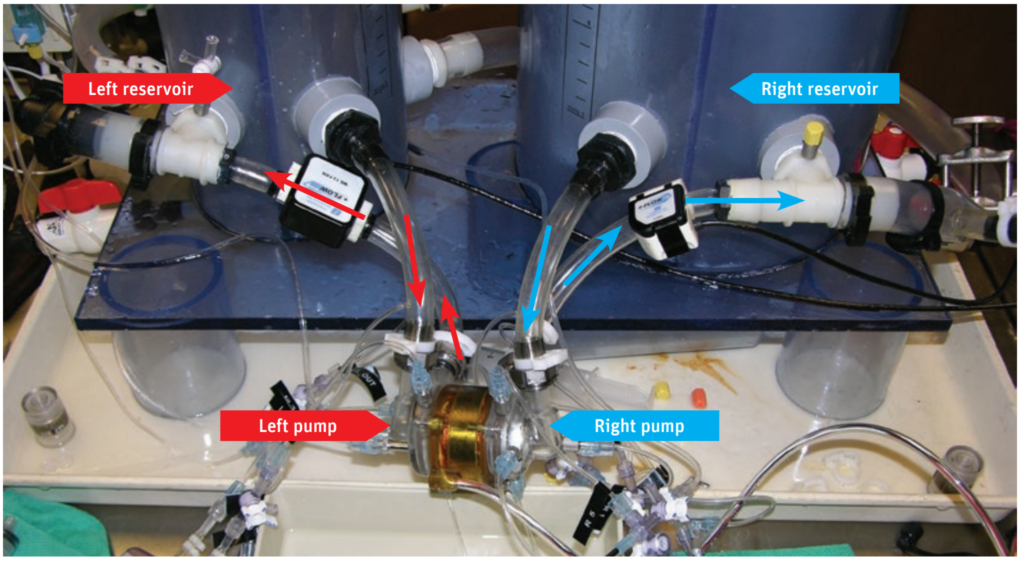
Researchers are currently working to validate the CFTAH design in preparation for in vivo testing in animals. Simulation is needed to capture data that cannot be collected during physical testing as well as to evaluate design alternatives in less time and at a lower cost than could be accomplished with physical testing. Multiphysics simulation was required because of the importance of both electromagnetic forces and fluid flow in determining pump performance. SimuTech engineers began by developing a three-dimensional electromagnetic model in ANSYS software to predict the magnetic radial and axial forces and torques acting on the rotor for different axial and radial offset positions of the rotor with respect to the stator. The final electromagnetic model contained 780,000 hexahedral elements, and the team performed a mesh sensitivity study to validate its accuracy.



▲ Electromagnetics model of the motor. The rotor is shifted both axially and radially to determine the force system acting on the rotor for arbitrary rotor position. Severe localized saturation occurs at the overhanging rotor end.



▲ 3-D flow simulation of rotor speed contours

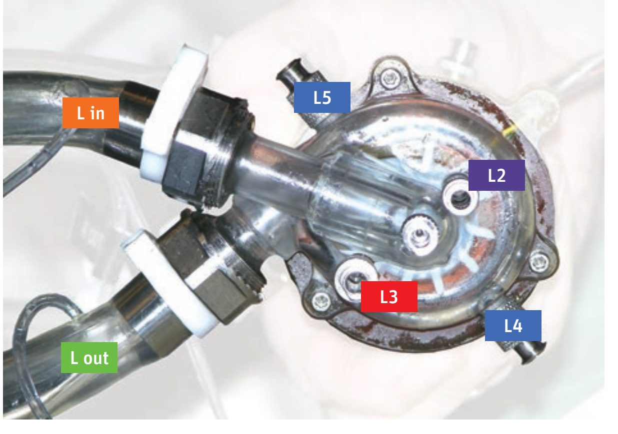


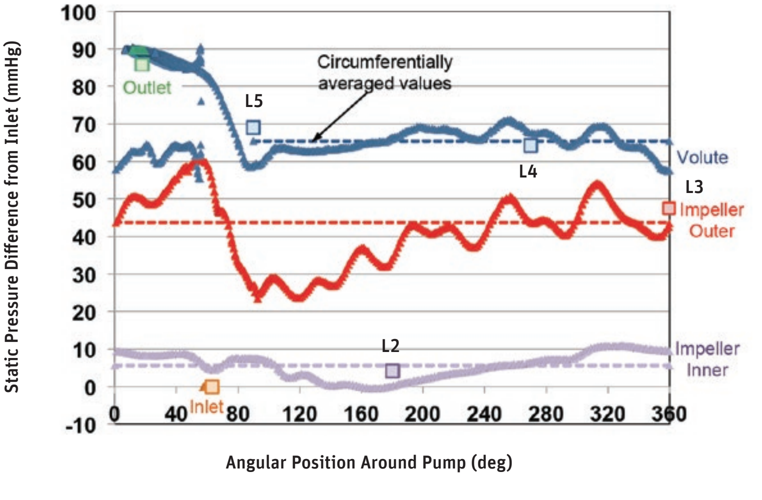
▲ CFTAH prototype connected to in vitro test loop

Electromagnetics simulation was used to determine the radial and axial forces generated by the magnet that moves the rotor toward the centerline position as a function of the offset from the center. The design depends on restoring forces to help control and limit the position of the rotor. The simulation showed that the magnet produces a linear radial restoring force of approximately 1,500 Newtons per inch of offset from the centerline of the bearing.

MODELING FLUID FLOW

Engineers used the results of the electromagnetic simulation to create a magnetic force table that they incorporated into an ANSYS CFX computational fluid dynamics (CFD) simulation as a user-defined function. The team then used CFD to model the fluid flow through only the journal bearing region of the pump to calculate the rotor radial position at various rotor speeds. As 95 percent of the radial forces are generated in the bearing region, this approach provided accurate positioning results without the need to model the entire pump assembly. Two different hex meshes of the bearing region were used to ensure that the results were independent of mesh density. The finer mesh had 528,000 elements, and the gap between the rotor and housing was 11 elements thick, while the coarser mesh had 216,000 elements and an 8-element-thick gap. The fluid was defined as a water/glycerin mixture with





▲ Multiphysics simulation vs. physical testing: Surface static pressures show good correlation.

The SimuTech Group is performing multiphysics simulation that incorporates electromagnetics simulation coupled with fluid flow to fully explore the operation of the artificial heart.

		ROTOR TORQUE			RIGHT PUMP APERTURE SIZE			ROTOR AXIAL FORCES			
Rotor Speed (rpm)	Q - AoP - PAP	CFD (oz* in)	Test (oz* in)	Diff. %	CFD (in)	Test (in)	Diff (in)	Hydraulic (N)	Magnetic (N)	Imbalance (N)	Imbalance Δp (mmHg)
2200	3 - 70 - 20	1.71	1.60	7.0	0.027	0.035	-0.008	-0.33	0.089	-0.24	-2.9
2800	6 - 90 - 20	2.89	2.65	9.0	0.037	0.046	-0.009	-0.35	0.014	-0.33	-4.1
2800	6 - 90 - 30	2.87	2.65	8.2	0.045	0.048	-0.003	-0.32	0.023	-0.30	-3.6
2800	6 - 90 - 40	2.85	2.68	6.4	0.055	0.061	-0.006	-0.57	0.092	-0.48	-5.8
3400	9 - 110 - 30	4.37	4.37	0.0	0.053	0.062	-0.009	-0.22	0.089	-0.13	-1.6

▲ Multiphysics simulation vs. physical testing: Surface static rotor torque, right pump aperture size and rotor axial forces show good correlation.

density equal to blood to match the in vitro test conditions. The model was evaluated at three different rotor speeds.

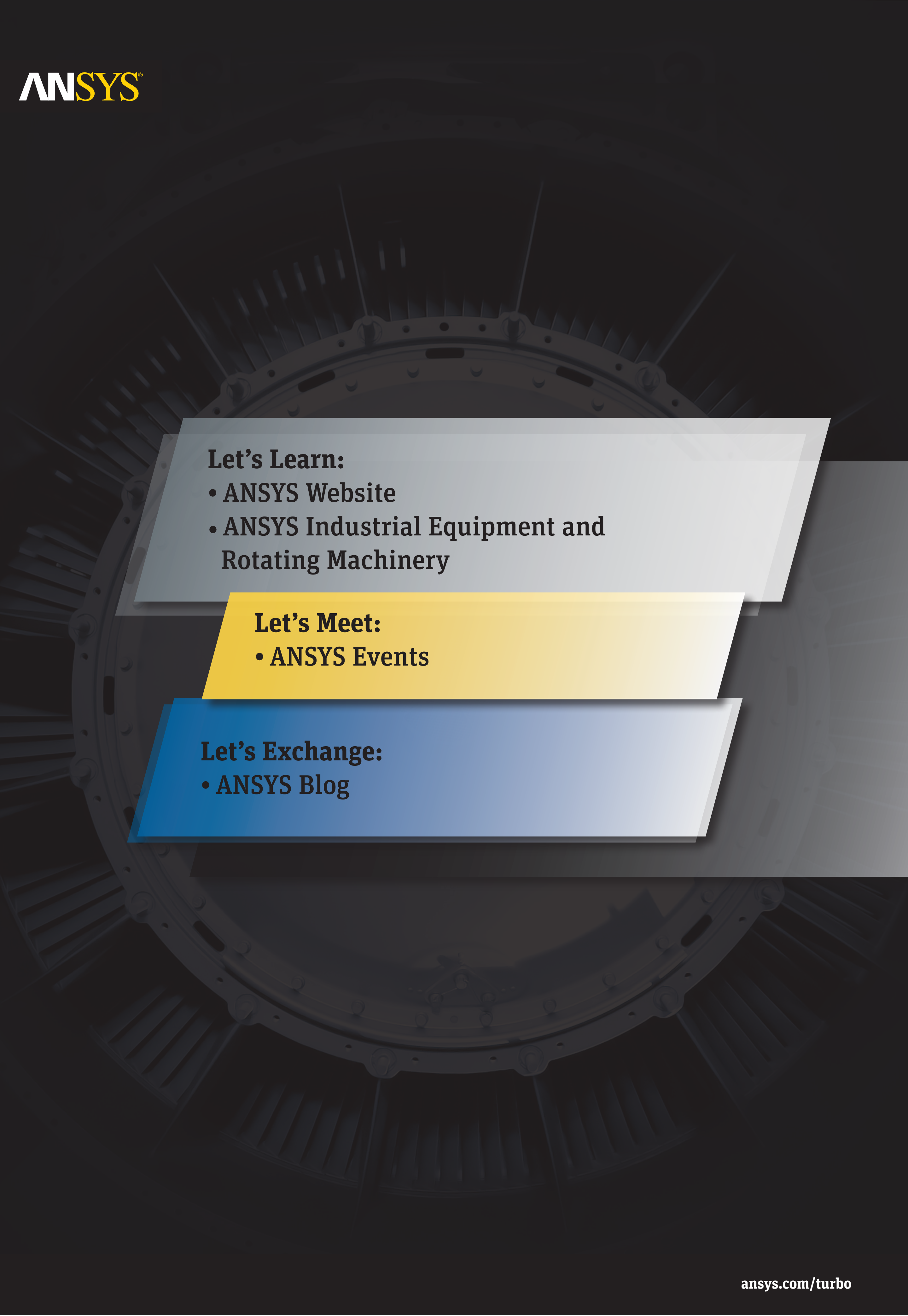
Deformation of the domain as the rotor moves radially was accomplished by using a moving mesh approach in which displacements relative to the initial mesh were specified with a user-defined function. A diffusion equation representing rotor displacement was included to determine mesh displacements throughout the remaining volume of the mesh. The magnetic restoring forces due to rotor movement were compared with hydraulic forces predicted with CFD to determine the force-balanced rotating assembly position.

RESULTS MATCH PHYSICAL TESTING

For the next step, the team modeled the complete CFTAH pump assembly and compared its performance to physical test results. The full three-dimensional pump model consisted of approximately 15 million elements, including tetrahedral, prism and hexahedral elements. Engineers ran the simulation on a 12-node high-performance computing platform. Due to symmetry and blade clearance in the volute regions, researchers used a frozen-rotor multi-frame-of-reference model and fixed the rotating assembly in one blade orientation — that is, the flow was modeled under steady-state conditions. They used the same water/glycerin mixture as the fluid and employed the k-omega shear stress transport turbulence model. The model was evaluated at three different volumetric flow rates and three different rotational speeds spanning the intended range of use. Engineers positioned the rotating assembly at the force-balanced radial location calculated earlier and moved the assembly iteratively to an axial location that yielded a right pump outlet pressure matching the in vitro test data.

Multiphysics simulations predicted hydraulic performance, surface static pressures throughout the pump, and rotor torque within 5 percent to 10 percent of the prototype's measurements. Radially, the rotating assembly hydraulic forces balanced with the magnetic loads within 5 percent. The axial position of the rotating assembly predicted by simulation matched experimental measurements within 0.25 mm. An axial force imbalance of 0.1 N to 0.5 N toward the left pump was found across the pump's operating range. This force corresponds to a static pressure difference of 2 mm Hg to 6 mm Hg. The reasons for this imbalance will be examined further in future simulations. Overall, these results are quite good and well within the expected level of agreement for this phase of the program.

In a more sophisticated simulation model under development, the team defines the rotor as a moving mesh and uses electromagnetic and hydraulic forces to move the rotor into a force-balanced position during the simulation (instead of setting the initial radial and axial positions). This model will use blood as the fluid, enabling examination of the shear forces exerted on the blood by the pump's surfaces. Shear forces need to be closely controlled; if they are too high, the blood cells may be damaged. On the other hand, if shear forces are too low, the blood may clot. The more-sophisticated simulation model will play an important role in finalizing the design of the CFTAH as it moves into animal testing and toward eventual human use.


ANSYS
Let's Learn:
• ANSYS Website
• ANSYS Industrial Equipment and Rotating Machinery
Let's Meet:
• ANSYS Events
Let's Exchange:
• ANSYS Blog
ansys.com/turbo