ANSYS®

ADVANTAGE™

Excellence in Engineering Simulation

2012

Special Issue:

OIL AND GAS

DESIGN WITH CONFIDENCE

Energy companies use ANSYS software to design equipment and develop new technologies across the entire oil and gas supply chain.

By Ahmad H. Haidari, Global Industry Director
Energy and Process Industries, ANSYS, Inc.

Increased production and development of new resources in the oil and gas industry are driven by economic viability and technology breakthroughs. Consequently, investments in R&D and capital expenditures are growing to support production from old fields, remote locations and unconventional resources. To accelerate such development, oil and gas engineering teams leverage high-fidelity engineering simulation solutions combined with existing practices. Virtual engineering usage has broadened as a result of increasing project complexity and continual advances in electronics, electromagnetics, fluids, thermal and structural mechanics simulation technologies.

Simulation-Driven Product Development encourages upfront modeling and simulation to drive entirely new solutions, evaluate multiple design variations and accurately predict performance in real-life situations. By using engineering simulation earlier in the design cycle, R&D teams can perform design optimization and parametric analysis using virtual prototyping long before manufacture and deployment. The practice can lead to simple and well-designed equipment that meets quality, durability and dependability requirements while shortening project time and reducing development cost.

The oil and gas industry has used simulation for exploration and reservoir engineering for decades. Today, a growing number of companies are taking advantage of the benefits that high-fidelity engineering simulation brings to drilling, completion, production, processing, storage, transport and refining. Beyond providing product and process development tools, simulation helps an organization to be more productive, making good use of its engineering knowledge and offering efficiency at a time when the pool of qualified engineers is shrinking. Furthermore, engineering simulation combined with data management helps to capture cross-domain, geographically distributed practices.

A systematic use of these capabilities, especially earlier in the design process, enables collaboration among distributed engineering centers in developing, gaining and managing more via computer resources. The necessary IT infrastructure typically includes high-performance computing (to accommodate robust design) and engineering knowledge management (EKM) to ease collaboration, work flow, knowledge capture and dissemination.

TABLE OF CONTENTS

4
Model+Make
Scott Parent, vice president of technology at Baker Hughes, provides insight on how a systems engineering approach and simulation-driven design improve product reliability.

8
Making the Connection
A challenging offshore oil pipeline application leverages simulation to check structural load conditions of an inline sled.

11
Deep Thinking
ANSYS structural mechanics helps save years in designing the first steerable conductor for enhanced oil recovery.

14
Forecasting Underwater Noise
Simulation soundly predicts hydro-acoustics during offshore pile driving.

16
Tracking Down Vibrations Fast with FSI
With high-speed iterations between mechanical and fluids software, fluid–structure interaction quickly pinpointed the cause of damaging vibrations and assessed new designs for offshore oil and gas equipment.

18
Catch the Next Wave
Hydrodynamic simulation helps to deliver two- to three-times wave power efficiency improvement.

As installation and operation intensify in the Arctic, in deeper waters and at more remote sites, engineering challenges become more complex since equipment needs to be highly sophisticated and operate reliably — and often remotely. Much oil and gas equipment is increasingly intelligent, driving a complicated composition of hardware, electronics and software. A new systems-level approach is required to manage the shift from single-component and single-physics design to multicomponent, multiphysics engineering. Advanced simulation solutions from ANSYS are meeting these emerging industry requirements. Energy engineers can simulate across various physics domains, including structural mechanics, fluids, thermal and electromagnetics. Single users and teams can focus on individual components, subsystems or an entire product system, sharing results and collaborating seamlessly.

The breadth of capabilities and the extent to which complex problems can be modeled in an integrated environment make it appear as if these solutions are targeted for use by highly technical scientists in R&D labs. Though analyst usage continues to grow, simulation is increasingly being used to drive more value out of existing engineering teams across all departments. ANSYS helps to achieve broader deployment of engineering simulation through a unified collaboration and simulation platform called ANSYS Workbench. This environment helps companies to work efficiently, using simple tools to define and manage work processes via drag-and-drop workflow and customization. Connections with other CAE and CAD tools enable engineers to work in heterogeneous PLM/IT environments.

ANSYS is unique in that it provides engineering simulation software for the entire range of the oil and gas supply chain's design and analysis requirements. Solutions include hydrodynamic, fluids, structural, electromagnetics (low- and high-frequency), electronic cooling, code check and sea-keeping applications. The full portfolio enables oil and gas leaders to engineer their entire system across disciplines and within the full range of engineering complexity.

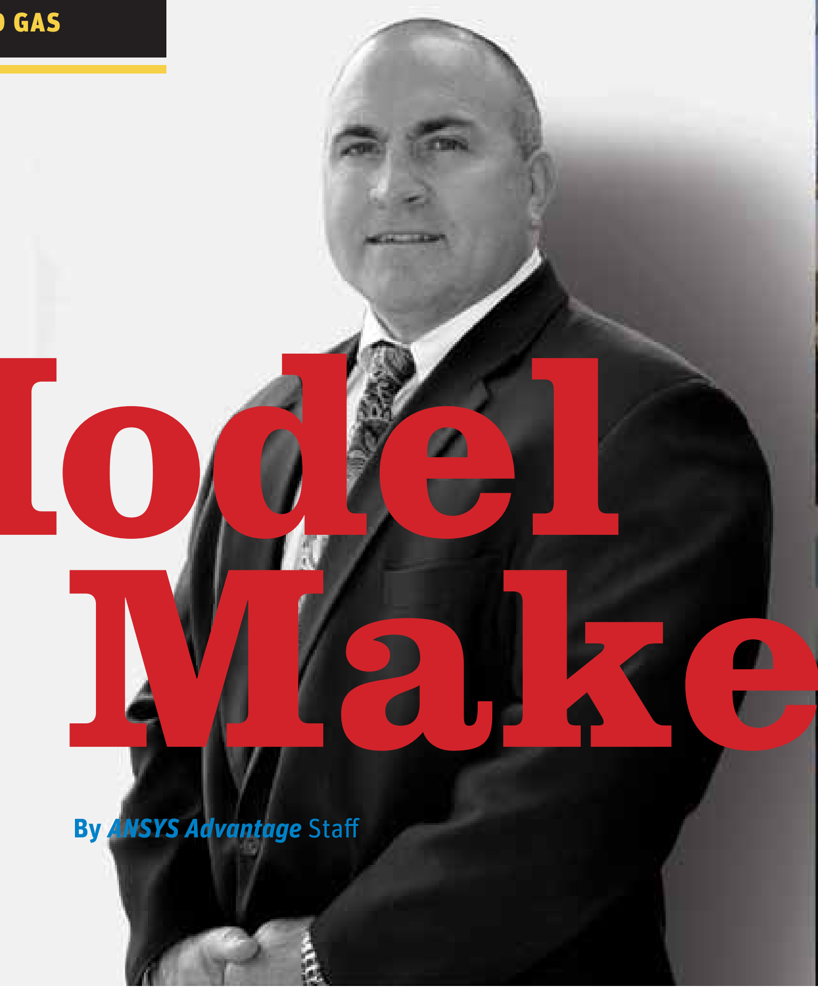
This special issue of *ANSYS Advantage* includes many customer case studies to inspire you to maximize your own use of engineering simulation. Follow their examples to master the complex challenges you face today.

In "Model+Make," Scott Parent at Baker Hughes describes how a collaborative, system-level approach is driving substantial gains by improving reliability, accelerating product development and reducing risk.

Other articles in this special edition demonstrate the range of engineering problems that can be solved, and provide examples of how simulation accelerates engineering design processes and increases product reliability. Sophisticated products and complex applications push the limits of engineering and advanced simulation technologies help to meet those challenges.

In these pages, you'll learn firsthand how well-established design processes, upfront simulation and validation continue to create greater confidence in even the most groundbreaking innovations — enabling oil and gas companies to gain insight and manage risk so that these organizations can invest in complex projects.

With a global presence, industry-specific experience and comprehensive multiphysics portfolio, ANSYS helps energy companies engineer an entire system across multiple disciplines, geographic locations, and product components and subsystems — preparing them for the continued engineering challenges the future will surely bring.



Model + Make

By *ANSYS Advantage* Staff

Scott Parent, vice president of technology at Baker Hughes, provides insight on how a systems engineering approach and simulation-driven design improve product reliability.

Baker Hughes is one of the world's largest oilfield services companies. It provides products and services to international oil companies (IOCs), national oil companies (NOCs), and independents both large and small. Baker Hughes also manages more than 3,000 drilling and production rigs, providing consultation, expertise, equipment and planning.

Scott Parent, vice president of technology at Baker Hughes, leads more than a thousand engineering and science professionals in three major technology centers in the United States and Germany. He also coordinates efforts at two research and engineering organizations in India and Russia. The innovative tools and systems these groups develop incorporate a broad array of fundamental science — including materials, nuclear, resistivity, acoustics, resonance imaging, ballistics and fracture mechanics — as well as traditional electrical and mechanical engineering. Parent has spent more than 20 years in product R&D in automation and robotics, manufacturing, real-time optimization, aircraft engine design and reliability modeling, clean coal technology, and, most recently, products and services for the energy and oil and gas industries. Having used ANSYS products throughout his career, he talked with *ANSYS Advantage* about the importance of engineering simulation in the oil and gas industry.

WHAT ARE YOUR KEY TECHNICAL AND BUSINESS CHALLENGES?

Baker Hughes' major efforts are fairly straightforward: satisfying our customers' needs — including risk mitigation — and developing new products. As a result, we make significant investments in technology development, looking for advanced solutions. Some innovations Baker Hughes is currently working on include drone tools, manless rigs, and sophisticated sensors for high-temperature formation and fluid evaluation. These highly engineered products require a systems engineering approach, rather than the silo one traditional to our industry.

IS THIS MOVE AWAY FROM SILO ENGINEERING SIMILAR TO WHAT HAPPENED IN THE TELECOMMUNICATIONS MARKET?

Yes, this is a great opportunity for us to develop new methods and engineering tools. We need to build new infrastructure, and, clearly, more upfront planning and simulation will help. Baker Hughes has an incredible amount of science knowledge and homegrown simulations across engineers' and scientists' desks, including nuclear, ballistics, electromagnetics, acoustics, imaging, automation, vibration,

electronics, mechanics, thermal and resistivity. My commitment is to bring activities at all of our facilities together by applying toolsets and improving design standardization methods. Specifically, we have created several new analysis teams that bridge multiple product lines. Historically, these products operated together as a final product in our customers' wells, but their subsystems were rarely modeled together — until now. In the first six months of integrated simulations, our team has directly improved operating reliability of legacy products and impacted the successful launch of a new product.

WHAT ARE THE SUCCESS FACTORS FOR PRODUCT DEVELOPMENT?

Successful product development requires numerous things: good project management, talented systems-level thinkers, complex modeling, robust simulation processes and enough computing power to handle it all.

I think our education system trains engineers in a focused way, but we need electrical engineers, in particular, to think more broadly — more like systems engineers. In contrast, chemical engineers seem more likely to have this systems-thinking skill set, possibly due to the amount of formal process engineering education they receive in undergraduate and graduate school. Great simulations are well planned, and with the right systems-thinking team on task, these larger simulations can be performed successfully.

For our simulation process, we use design for six sigma methodology, which includes modeling, testing and analyzing. We start with a systems model and then flow down the requirements. As we step up the process, we become more reliant on increased computing power, which we can, thankfully, accomplish at a relatively low cost. Boosting computing power has become the backbone for complex modeling. It enables us to engage in more simulation-driven product development. Currently, we are partnering with

The software and other measures we have put in place have helped us to improve reliability and reduced our drilling systems nonproductive time by 25 percent globally in the past 18 months.

Microsoft® to explore the benefits of cloud computing. Some of the first platforms we plan to integrate are ANSYS software. We use ANSYS CFD with the high-performance computing option, allowing expedited solution time.

WHAT OTHER ANSYS PRODUCTS DO BAKER HUGHES' TEAMS USE?

In addition to ANSYS CFD (ANSYS Fluent and ANSYS CFX), we use the ANSYS structural mechanics suite. These programs enable structural and more-sophisticated analyses all in the same environment, providing the opportunity to improve technical communication across all levels of engineering and design. This integrated approach is unique and offers a full-service, soup-to-nuts approach that is important in testing our solutions in the complex and rapidly changing environment in which we find ourselves working today.

We also use ANSYS software for electromagnetics. We'll soon expand into ANSYS Maxwell and ANSYS Simplorer for low-frequency applications, ANSYS HFSS and Ansoft Designer for high-frequency applications, and ANSYS Icepak and ANSYS SIwave for electronic thermal management. With Simplorer and its capabilities to cosimulate with Fluent, HFSS, Maxwell and ANSYS Mechanical, we can build high-fidelity systems-level simulations that make physics and engineering considerations explicit. Because of the seamless connection between ANSYS products, we plan to use ANSYS DesignXplorer and ANSYS nCode DesignLife to ensure reliability, sensitivity and six sigma design, as well as to estimate the life of the simulated tool or component.

HOW DO ANSYS PRODUCTS HELP YOUR MULTIGENERATIONAL MODELING EFFORT?

Now that we can prove the importance of validation, verification and model maturation to our colleagues and the industry, everyone is beginning to recognize the importance of upfront analysis and systems validation.

Validation means that you understand the physics of the design before you build the circuit and test it, for example. As we use field data to validate complex models created with ANSYS products, correcting for missing items or calibrating for issues, we create a record of knowledge, a record of the application's physics. This is our definition of a multigenerational model. When a model has been validated and then proven in this industry, it becomes a valuable asset — a company jewel. Today, we track our models manually, but we are looking at tools to manage this process for us.

For example, we are considering the ANSYS EKM tool to help us index, manage and track our engineering models. What we like about EKM is its capability to quickly search past and current simulations to find appropriate files and extract knowledge in an efficient manner. With EKM, we would not have to repeat a simulation if someone has done it in the past. When an engineer leaves the project, the software stores a copy of all of the simulations, making the files accessible to other team members around the globe.

HOW HAS PRODUCT DESIGN CHANGED AT BAKER HUGHES?

Three years ago, our testing processes consisted primarily of field testing. We designed it, we built it, and we took both new products and systems into the field with less attention paid to lab testing or validating of upfront models.

But in reality, our products must operate with other systems and tools. Before we begin design, we need to understand the operating or application environment, how our products integrate with other products, and the physics of final deployment. We must look beyond a single tool and develop a much better systems design up front.

At Baker Hughes, we have restructured into cross-disciplinary teams that work together to consider submodels, integration and parametric methods. When a team begins to develop a new product or solve an engineering problem, they ask questions such as: Do I have something new, or am I doing something we've already solved? What is my goal — to do something new or to make it better? To make something last longer? It's important to realize that you don't always start with a former product. In many cases, it doesn't save us any time. Instead, we can design a new platform using a new set of tools.

Now I am working to encourage our teams to use simulation earlier in the process. For example, it isn't optimal to test a noisy motor to find out what the problem is. It is far less expensive in time and money to model the motor before building it and avoid the noise in the first place. Validated models enable us to improve first-pass reliability and reduce total time to market.

How do we reduce risk? We calculate it up front in simulation stages at both systems and component levels.

ANSYS allows us to try it before we build it.

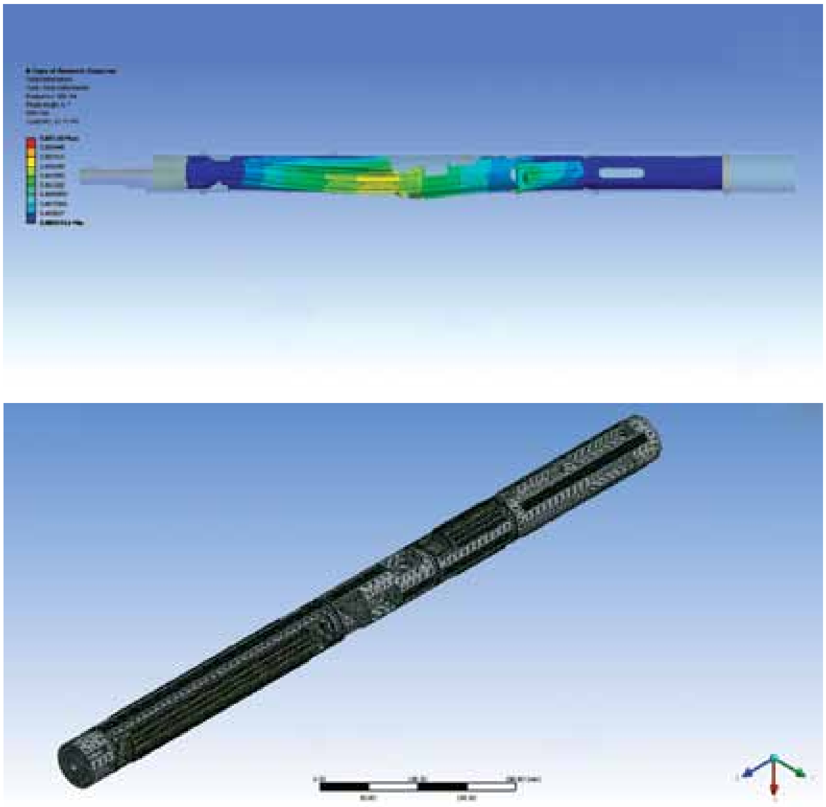
CAN YOU SHARE AN EXAMPLE OF A RECENT DESIGN?

One task was to create complex simulations involving the bore hole (down-hole environment) in drilling systems deployment. The drilling environment exposes our tools to high-temperature and high-vibration conditions. We needed to better understand how these conditions might change tool performance, and specifically how they contributed to catastrophic failure of embedded electronic circuit boards.

Accessing data collected around root-cause electronics failure (primarily high vibration and temperatures for extended periods), we then used ANSYS products to simulate tools and components in this down-hole environment. Our aim was to improve survivability of the printed circuit board assemblies. By analyzing component mounts, electronics packaging and board mounting scheme in the down-hole tools, we reduced the failure rates of these critical subsystems significantly.

HOW DO YOU ADDRESS RELIABILITY?

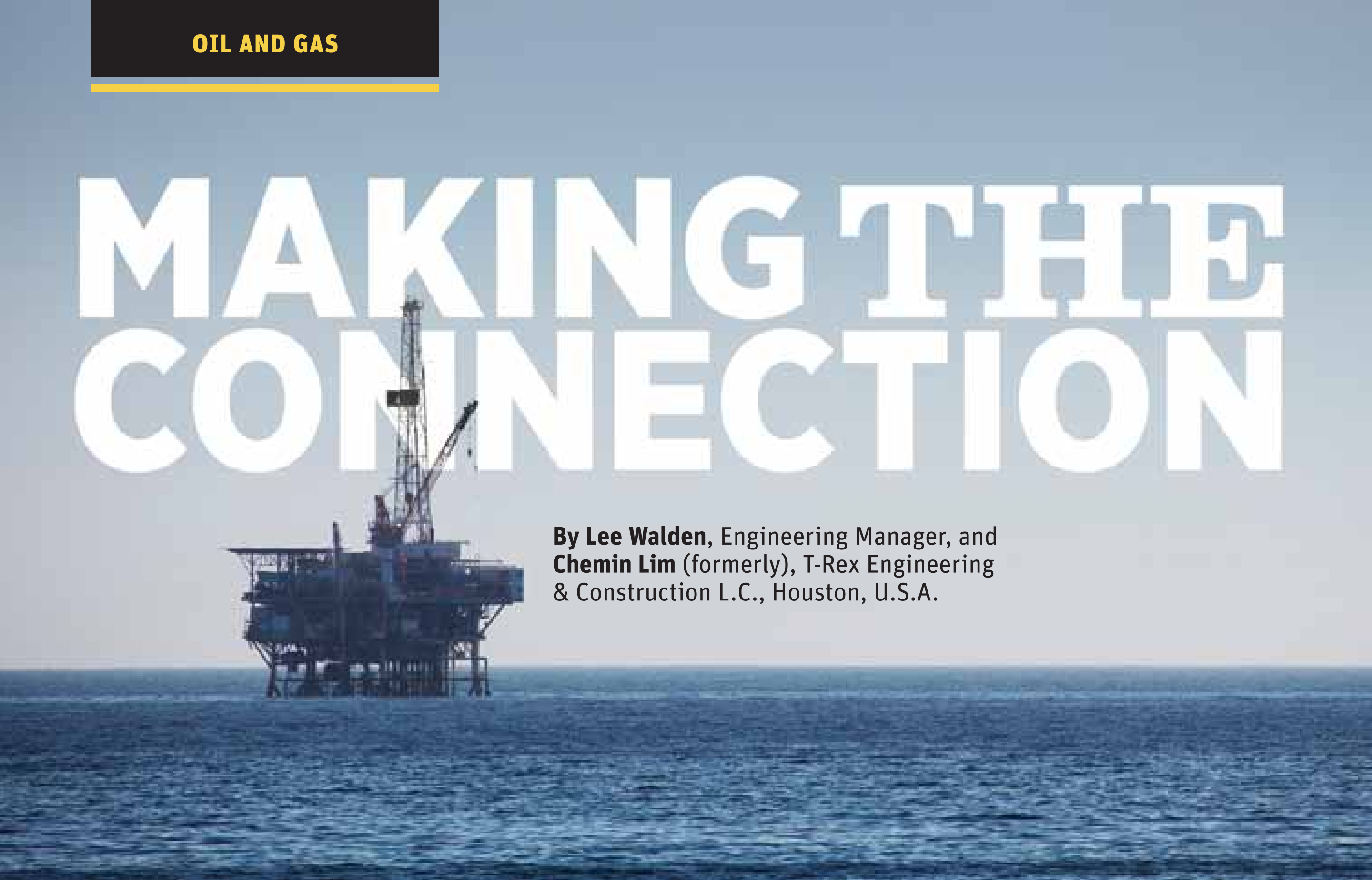
The complex simulations using ANSYS software allow us to improve product reliability. In our industry, nonproductive time on a rig is measured in hundreds of thousands of dollars. It is critical that we continually improve our design capabilities for reliability and manufacturability. ANSYS software enables us to consider these features in the context of trading off design features with performance criteria. We use these capabilities more and more to enhance lifecycle costing, including repair and maintenance, through robust design and failure analysis. The software and other measures we have put in place have helped us to improve reliability and reduced our drilling systems nonproductive time by 25 percent globally in the past 18 months.



Just one way ANSYS structural technology is used at Baker Hughes is to simulate tools and components in a down-hole environment to improve survivability of the printed circuit board assemblies. This simulation demonstrates high-frequency dynamic deformation (top) and the integrated FEA mesh (bottom) of a down-hole tool.

OVERALL, HOW DOES THE USE OF ANSYS TOOLS IMPACT YOUR DESIGN PROCESS?

"Model and make vs. build and break" is what I call our initiative around this. Simply put, ANSYS allows us to try it before we build it — to try things on the drafting board and exercise component and system models as though they were in the lab or field application. We can combine multiple scientific and engineering disciplines earlier in the process, creating an environment in which development can evolve creatively and economically. ANSYS tools — and simulations in general — are helping Baker Hughes to take a more aggressive approach in our design processes. We can make decisions earlier in that process, and, with a broader interdisciplinary talent pool, design with increased confidence. Identifying and resolving issues early in the process helps us to mitigate risk, save millions of dollars, reduce development time, and drive customer value. Ultimately, this approach drastically improves first-time yield of products coming from R&D.



MAKING THE CONNECTION

By **Lee Walden**, Engineering Manager, and
Chemin Lim (formerly), T-Rex Engineering
& Construction L.C., Houston, U.S.A.

A challenging offshore oil pipeline application leverages simulation to check structural load conditions of an inline sled.

The offshore oil exploration and drilling industry strives continually to develop new subsea technologies to meet the rising demands for petroleum products. Since most of the "easy" fields have been tapped, harvesting distant offshore oil becomes more challenging because the pools are situated under thousands of feet of water.

Subsea technology covers a wide range of offshore activities. One main subsea technology is a pipeline system — sometimes more than several hundred miles in length — that transfers oil and gas products from the seabed to other destinations. The pipeline consists of various mechanical, electrical and hydraulic parts that are supported by several subsea structures.

INLINE SLED

A major component of this subsea system is the inline sled (ILS), a pipeline support structure that allows a future pipeline tie-in to be made quickly and efficiently on the sea bed. The sled is dropped over the end of a vessel's stinger — a specialized piece of equipment that is mounted onboard a ship — along with miles of piping. The pipelines are welded together on the stinger to facilitate the process of subsea installation.

The ILS comprises a mudmat platform (ILS foundation module) and a frame system that supports a wye block (a fitting that joins pipelines), branch piping, transition piping, valves, and an end hub support that is integrated into the pipeline. The main oil flows from the right (as shown in Figure 1); the future tie-in oil flow comes from the hub and joins at the wye block. The valves control the oil flow, and the hub is the open connection for future pipeline connections. A tapered transition of pipe is installed at each end of the sled's piping system to resist bending moments caused by the ILS going through the stinger.

SURVIVING CHALLENGING CONDITIONS

The engineering challenge is to design the ILS so it survives under 7,000 feet of seawater, sustains severe environmental loads and resists corrosion — all while minimizing the high risk of damage to equipment and hazard to human life during installation. T-Rex Engineering and Construction conducts studies to fully understand conditions where subsea structures will be constructed. The company's work includes fabrication, transportation, installation and operation. Based on extensive subsea experience, the engineering team collects all possible data to simulate the structure in real-world conditions. In fact, the organization has 15 years of experience in the development and design of subsea structures, all of which are still operating in the subsea field. T-Rex holds the world record for installing the deepest subsea structure.

A subsea structure experiences its worst load conditions during installation because the ILS is subjected to the weight of the suspended pipe (flow line) as well as the floating motion of the vessel. As the vessel lays the pipeline over the stinger, the ILS undergoes severe tension and bending loads at the top and bottom curvature of the pipeline (Figure 3a).

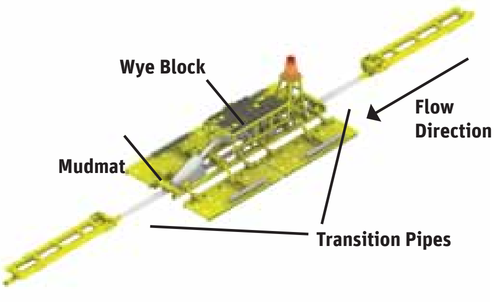


Figure 1. Inline sled structure

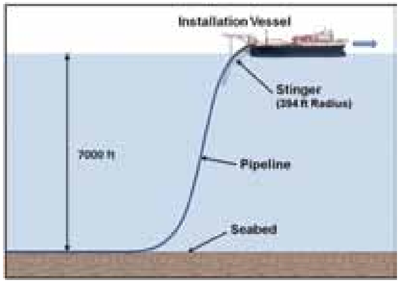
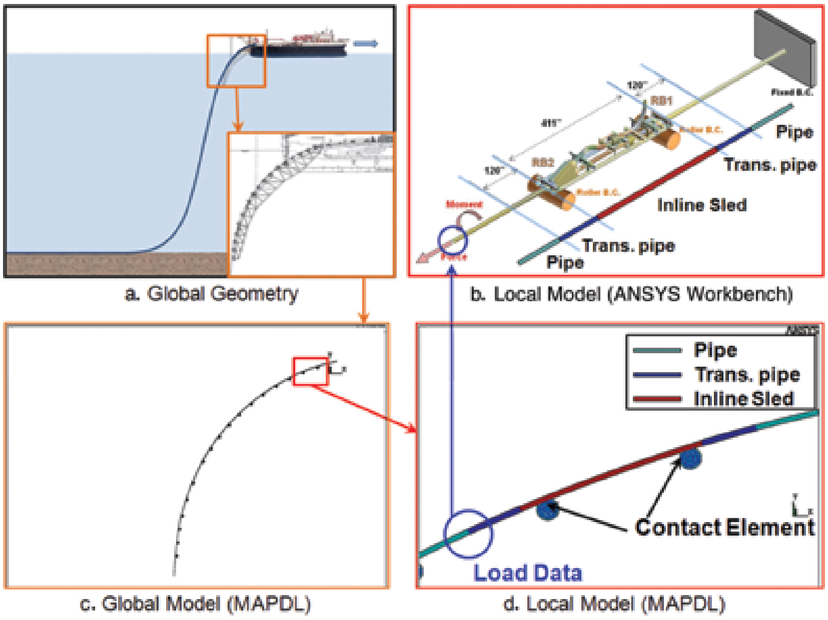


Figure 2. Installation of S-laying pipeline



a. Global Geometry

b. Local Model (ANSYS Workbench)

c. Global Model (MAPDL)

d. Local Model (MAPDL)

Figure 3. Analysis

T-Rex engineers determine the tension and bending load values to ensure a robust and safe design that will withstand the installation process. Analyses are performed to predict whether excessive stresses and deformation in the ILS system arise during the installation process.

SIMULATING THE SYSTEM

Simulations determine load conditions on the pipeline; they also help engineers design the ILS to handle that specific load. In one application, T-Rex engineers used ANSYS Mechanical APDL (MAPDL) to analyze a 2-D global model to determine these load conditions. They used ANSYS Workbench to apply these load conditions to the local 3-D solid model of the ILS. This type of systems modeling with ANSYS tools enables T-Rex to ensure the robustness of the design.

The team used beam elements to complete the 2-D global model of the pipeline and ILS, as shown in Figure 3c. To determine the beam element stiffness of the ILS, a separate 3-D solid model was simulated with ANSYS Workbench (Figure 3b).

For the 2-D global model, contact elements defined the contact conditions between the pipeline and the stinger's contact points, which are the group of bearing rollers (Figure 3d). Plane elements were used to model the rollers located on the stinger. This global model depicts the pipeline deformation on the stinger. The displacement load was applied at the end of the straight pipeline until the pipeline was in full contact with the stinger's roller boxes. To determine the local model's load condition — tension load and moment — reaction forces and moments were output at the end of the ILS on this global model.

The team used Autodesk® Inventor® 2010 to generate a detailed (local) 3-D model and directly imported it into ANSYS Workbench. The transition from Inventor to Workbench was smooth, and every component was imported without problems. The local 3-D Workbench model comprised 177,991 elements, including contact elements. Engineers used the sweep method to generate the mesh, and then the critical areas were refined. ANSYS Workbench automatically detected the contacting areas to generate surface-to-surface contact elements. Most of the contacting regions were defined by bonded contact behavior. The high-quality mesh produced in Workbench facilitated the convergence, calculation time and accuracy of results.

To simulate the roller box contact load conditions, frictionless support

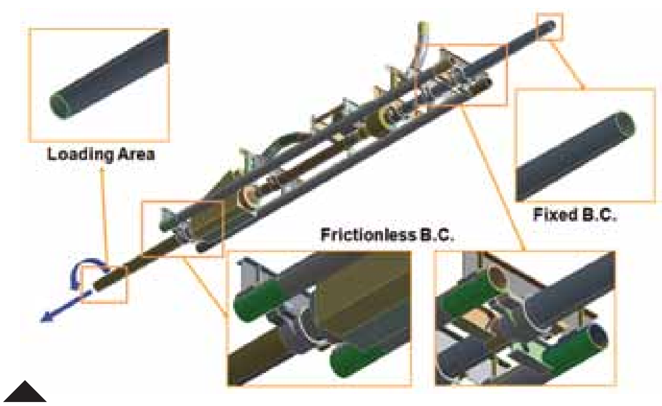


Figure 4. Boundary conditions

Using systems modeling with ANSYS tools enables T-Rex to ensure design robustness.

conditions (load tension/compression) were applied at both ends of the rail pipes, and the fixed boundary condition was applied at the opposite end of the structure. The load that was collected from the global MAPDL model was applied at the opposite end of the structure, as indicated in Figure 4.

As the design progressed, several components' geometries were changed, based on the stress results. For example, the connection between the pipeline and the ILS had a huge difference in stiffness, which caused a high stress concentration in that area (Figure 5a). At the end of this process, the new design reduced the peak stress by over 80 percent compared with the initial design (Figure 5b).

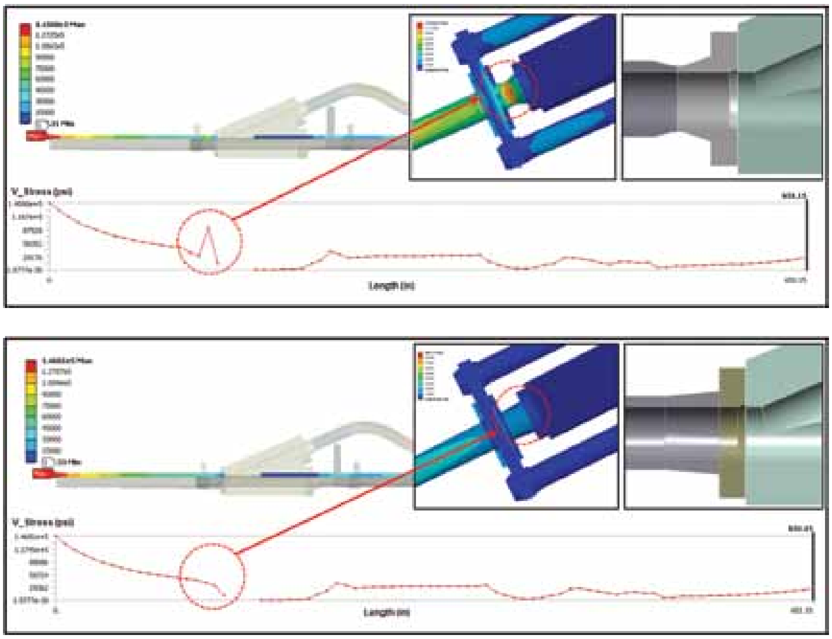


Figure 5. Initial model (top) and final model (bottom) of the connection between pipeline and ILS show stress contour through the inside pipeline. The new design decreased peak stress by over 80 percent.

ACCURACY ENSURES SAFTY

The combination of ANSYS Workbench and ANSYS MAPDL successfully simulated the field pipeline installation load conditions on this project. The analysis made it possible to obtain the exact load conditions for this complex geometry. It would have been almost impossible to obtain this level of accuracy required to improve the design without using ANSYS software products. This systems simulation procedure provides a wide range of solutions for pipeline installation process analyses. Furthermore, safety is an important factor. Subsea pipeline systems must be designed to be safely installed and maintained during oil production. The simulations in this application helped ensure that the subsea structure adhered to safety requirements.

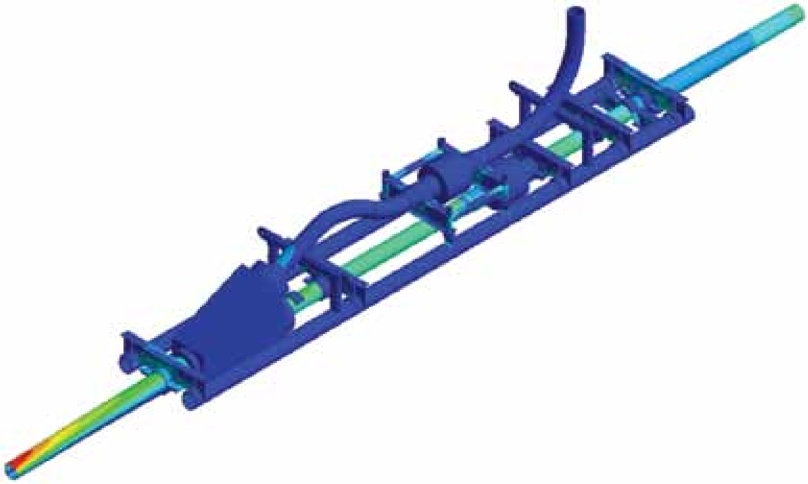


Figure 6. Von Mises stress contour

Deep Thinking

ANSYS structural mechanics helps save years in designing the first steerable conductor for enhanced oil recovery.

By Rae Younger, Managing Director, Cognity Limited, Aberdeen, Scotland

One of the biggest challenges in offshore drilling is accurate placement of the conductor casing. This component is a several-hundred-meter-long tube that is pile-driven into the ground prior to drilling to prevent mud from collapsing around the hole. At offshore locations, soils tend to be relatively soft with highly variable seabed properties; these factors contribute to accurate placement, since traditional conductors follow the path of least resistance.

Engineering consulting firm Cognity Limited has addressed this problem by developing a steerable conductor that can provide real-time accurate positioning. This device must withstand compressive forces of up to 600 tons as the conductor is pounded into the ground; it also must provide an unobstructed bore once it is driven to depth. Soils increase in strength with depth, which increases the moment and loads on the conductor as it is driven into the seabed. By using ANSYS Mechanical software in the ANSYS Workbench platform, Cognity engineers doubled the load-carrying capacity of the steering mechanism, allowing the conductor to be maneuvered in very deep soils. In addition, the team finalized the design in five months, a time frame months or possibly years less than would have been required using traditional design methods.

In drilling, each conductor must be positioned accurately to help maximize field production. For example, conductors might be spaced along a 2.5 meter grid at the platform with the goal of driving them into the seabed at an angle, spreading out to cover a predefined area. Since the drilling process weakens the soil, new conductors are naturally drawn toward existing wells — which might result in abandoning the conductor if it veers too close to a live well. Poorly positioned conductors, known as "junked slots," can result in a production company incurring lost time and additional expense in sidetracking them. A worst-case scenario can occur if a conductor is placed so close to an existing well that the

Rendering of steerable conductor

Image © iStockphoto.com/sculpies

milling tool removing the shoe — the blunt nose of the conductor — punctures a nearby producing well. Such a scenario may risk an uncontrolled release of hydrocarbons.

On behalf of a client, Cognity developed a fully steerable conductor capable of accurate placement in highly variable soil conditions. Over the past decade, the industry has trialed designs that passively vary the angle of the shoe in response to changes in soil conditions. But Cognity's design is the first to allow the conductor to be steered in real time from the drilling platform, which enables very accurate control of the final position. The benefits of such a system include possible increased production and reduced drilling costs through elimination of junk slots.

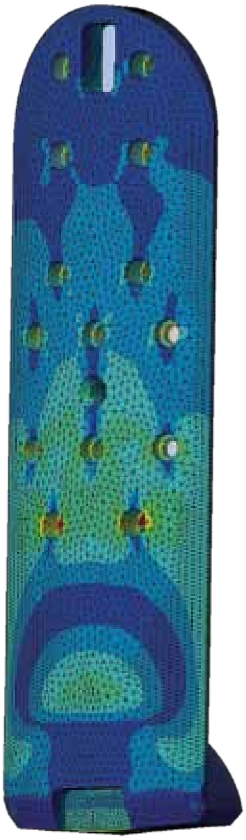
Nonlinear springs were used to represent soil forces acting on the conductor.

Design of the new steerable conductor presented major challenges: The most noteworthy is that the device must withstand the enormous forces required to drive a blunt object hundreds of meters into the soil. A traditional design approach would have required numerous full-scale prototypes, each tested to failure — a very expensive, time-consuming process. It would have taken several years for the Cognity team to develop a workable design; engineers would have had to settle for the first design that met minimum requirements rather than aiming to optimize the design.

Cognity took a different approach by using ANSYS Mechanical simulation software, developing virtual prototypes to evaluate alternative design performance. Cognity selected the ANSYS Workbench platform because of its ability to move new design ideas from computer-aided design (CAD) into simulation, then send proposed design improvements back to CAD — critical to meeting the project's tight time schedule. ANSYS Workbench offers bidirectional connectivity with popular CAD systems, including Autodesk® Inventor®, which Cognity uses.

ANSYS Mechanical software is also more applicable to design and optimization than other finite element (FE) analysis packages that Cognity evaluated. For example, an engineer can set up contacts with a click of a mouse, and these contacts will automatically update when the geometry changes. This feature saved Cognity considerable time in developing the device, which involves large assemblies of moving parts with multiple contact faces. The ANSYS structural mechanics software also provided excellent scalability on nonparallel machines, which helped to support fast turnaround times required for development.

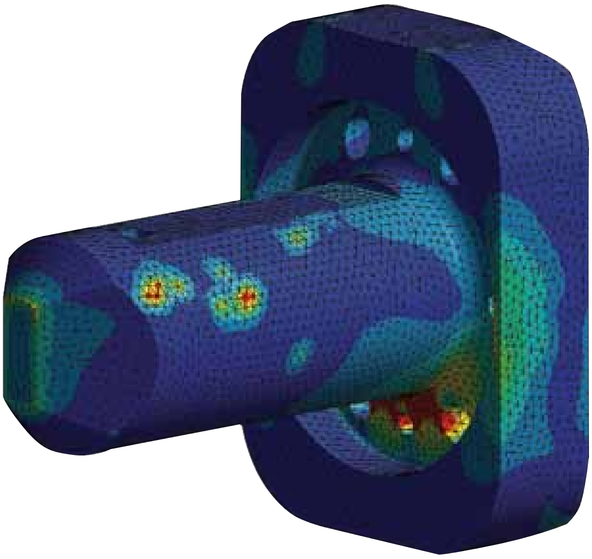
One factor critical to success was accurate modeling of the soil. Cognity engineers modeled various conductor concept designs and evaluated their performance when driven into a virtual environment: soil of varying properties. Soil has a highly nonlinear response, providing only compressive resistance under lateral loads. Friction acts on the outer surface of the conductor, creating drag forces that resist axial movement. Soil shear strengths vary with depth and specific location, and Cognity used actual soil test data to increase simulation accuracy. The engineers modeled the soil by using nonlinear springs connected to the conductor, tuned to provide the same stiffness as the soil at a particular depth. Mimicking soil, the nonlinear spring provides resistance proportional to the force up to its shear point; from that point on, the force is constant.



FE analysis results show stresses on the tendon.

One of the first tasks required was optimization of the conductor's shoe length. During drilling, the operator steers the conductor by changing the angle of the shoe. The shoe moves plus and minus 3 degrees in both x and y axes. A longer shoe better provides maneuverability in soft soil; however, it increases both the reaction force and resulting moment on steering components that connect the shoe to the rest of the conductor.

Cognity engineers modeled the conductor being driven into the ground with a 600-ton force from the hammer, then used analysis results to establish



Stresses on radial locking pads that hold HDH in place

the maximum generated moment and loads from the soil reactions at the shoe. This helped Cognity engineers to identify the loads on the critical steering assembly.

The next step was to apply these loads to the conductor's principle components so they could be optimized to resist the forces. One critical component is the hydraulic deflection housing (HDH), a 4-ton assembly within the 27-inch bore of the conductor. The HDH is responsible for holding the shoe in position and resists the forces generated by the soil. Analysis showed that the shock loading on this assembly is of the order of 150 g, which necessitated a 600-ton-capacity locking mechanism to hold the HDH in place. After the conductor is driven into the ground, the HDH is recovered, inspected and refurbished so it can be used again.

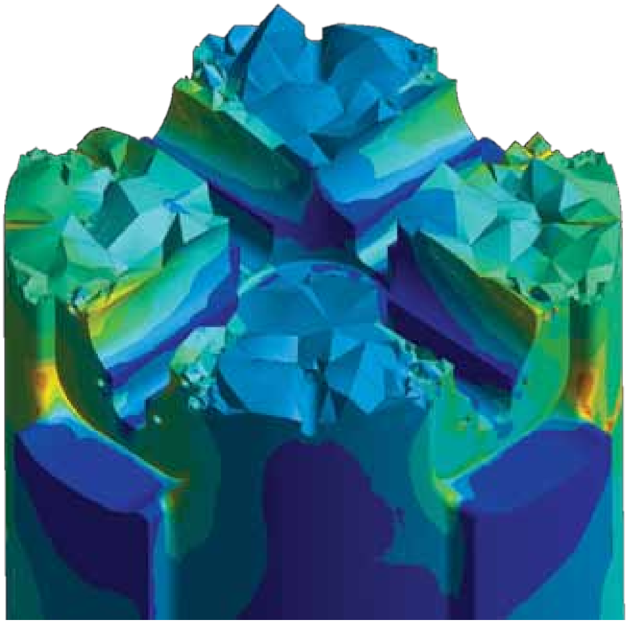
HPC Expedites the Design Process

The use of high-performance computing was critical to meeting delivery-time requirements of this project. Cognity runs structural mechanics software from ANSYS on a Dell® T7500 workstation with 12 cores and 24 GB RAM with RAID 0 SCSI drives for optimal disk speed. A typical model with about 750 K elements and many contacts can be solved in an hour or less, compared to about six hours without parallel processing. Parallel processing makes it possible to evaluate five to 10 design iterations per day, enabling Cognity to rapidly improve their design.

Cognity applied ANSYS Mechanical software to determine the stresses and deflections on the forging that makes up the HDH's body. The primary measure of its performance is its moment capacity, which identifies the ability to generate side load at an equivalent length. Engineers optimized the shape of the HDH, increasing its stiffness by adding material to high-stress areas and removing material from low-stress areas through an iterative process.

The HDH protrudes into the shoe; it is tapered to provide clearance for the shoe to move in both the x and y axes. Guided by structural mechanics analysis results, Cognity engineers found a more efficient way to taper the HDH and added supports in high-stress areas. As a result, the team was able to double the length at which the HDH connects to the shoe, effectively doubling the system's load-resisting capacity.

The original design used custom hydraulic cylinders that cost about $160,000 each and required four months for delivery. Using engineering simulation, Cognity engineers demonstrated that the custom cylinders could be replaced with the internal parts from off-the-shelf hydraulics that cost only $7,000 each and could be delivered within one month. For the overall project, Cognity was able to complete the design in only five months, approximately 70 percent less time than would have been required using conventional methods. ■



Stress analysis of the HDH helped Cognity engineers double system capacity by optimizing design.

MENCK hydraulic hammer

Forecasting Underwater Noise

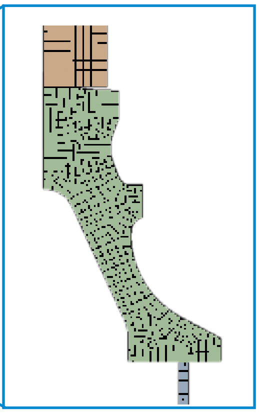
Simulation soundly predicts hydro-acoustics during offshore pile driving.

By Ulrich Steinhagen, R&D Project Coordinator, MENCK GmbH, Kaltenkirchen, Germany
Marold Moosrainer, Head of Consulting, CADFEM GmbH, Grafing, Germany

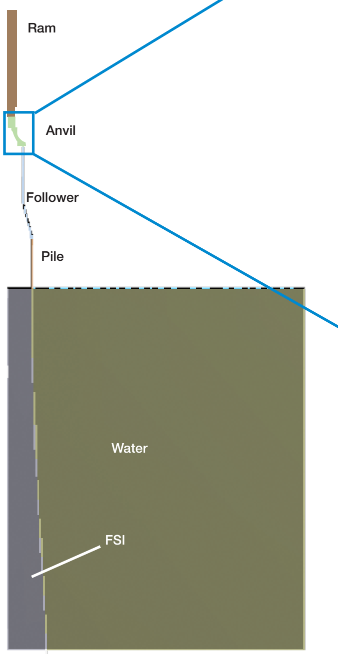
Across the vastness of our planet's oceans, localized sounds originate from many different sources, both natural and man-made. The man-made sources — for example, from ship traffic, drilling, mining and sonar equipment — have significantly added to underwater noise in recent decades. In the growing offshore wind power industry, sound emission related to installation and operation, especially underwater, is a growing concern due to its potential impact on nearby aquatic life.

In most cases, foundations for massive offshore structures such as wind turbines are formed by driving piles into the seabed with hydraulic hammers. The German company MENCK GmbH has a long history of developing, manufacturing and operating such hammers in water depths up to 2,000 meters (1.25 miles). Predicting hydro-acoustics during offshore pile driving is, therefore, of great interest to installation contractors who must comply with tight sound-emission thresholds. For example, the regulatory limit in Germany for underwater sound exposure level at a distance of 750 meters from a construction site is 160 decibels (dB) at a reference pressure of 1 microPascal ($1x10^{-6}$ Pa) [1]. Knowing the sound emission prior to construction helps contractors to select and design noise protection systems — such as air bubble curtains or air-filled cofferdams around the pile — that will meet local project requirements [2]. Reducing underwater noise, however, remains an ongoing subject of research, as no single system is appropriate for all situations.

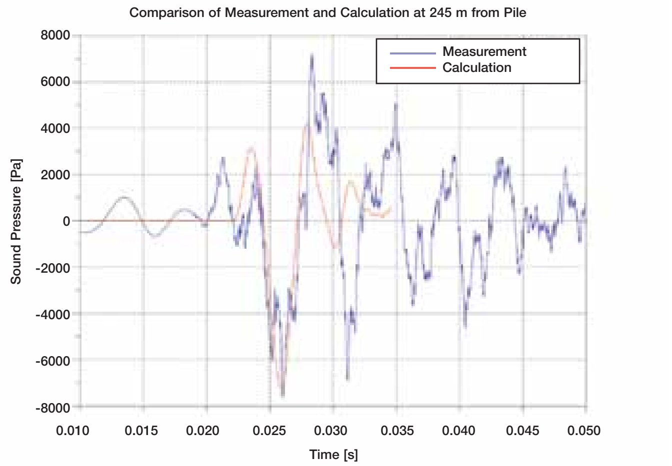
With this background, a MENCK research team initiated an application project with computer-aided engineering software and services company CADFEM to use ANSYS simulation tools to numerically predict underwater sound emission. Transient structural analysis of the driving impact is commonly performed to evaluate mechanical characteristics of highly loaded hammer components, such as the ram, anvil, adapter plates and followers. MENCK engineers expanded this original simulation to consider noise propagation by two-way coupling of the pile vibrations and water pressure using the acoustic elements in ANSYS Mechanical software.



Closeup of hammer's meshed region including lower section of ram, anvil and top of the follower



Two-D axisymmetric simulation model in ANSYS Mechanical software. Components of the hammer and pile system along with the water and seabed zones that they inhabit; zone of interest for FSI is indicated with a line.

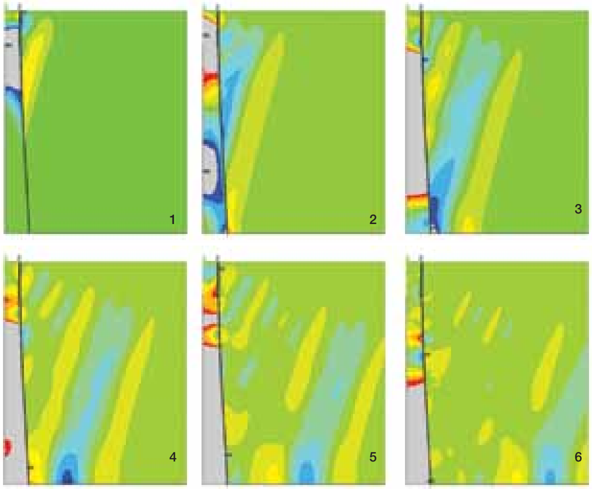


Comparison of measured and calculated underwater sound pressure at a distance of 245 meters from the pile. Knowing the sound propagation law for this region, the sound pressure at 750 meters can be calculated and converted into decibels (dB).

The two-dimensional axisymmetric simulation consisted of the ram, anvil, follower, pile, soil and water. Coupling the water domain to the structural elements of the pile was made possible via ANSYS Mechanical fluid–structure interaction (FSI). The MENCK team set the ram's initial velocity to achieve the desired impact energy, with all other components being at rest. Boundary conditions included acoustical reflection at the seabed and acoustical absorption at the outer boundary, while the elastic soil properties were modeled as lateral springs. Researchers applied a simple absorption boundary condition because, when only the short-term near-field sound propagation is of interest, reflection is not a crucial issue for transient analysis of acoustics problems. The team additionally approximated the water–air interface as a free surface with zero pressure, which is appropriate for a non-rigid boundary.

MENCK's engineers set up the simulation in the ANSYS Workbench environment using ANSYS Parametric Design Language (APDL) to control FSI and acoustics parameters. FLUID29 acoustic elements allowed modeling the sound field to cover modal, harmonic and transient solutions. The theory of acoustic waves underlying the FLUID29 element approach is based on the same fundamental equations as computational fluid dynamics (CFD): conservation of mass and momentum. However, assumptions — such as zero-flow velocity and inviscid, compressible fluid properties — were made that result in a linearized acoustic wave equation. A linear equation is reasonable because, even for very high sound pressure levels, the acoustic pressure variations generally represent only about 0.2 percent of ambient pressure.

In concert with the FLUID29 elements, additional displacement degrees of freedom supported the interface to the structural domain. In this way, MENCK could model the full coupling between the fluid (acoustic) and structure domains to account for the sound radiated by a vibrating structure and, at the same time, consider the additional load of this sound pressure field onto the structure [3]. The near-field solution in the vicinity of the pile could then be used to predict the sound pressure level in the far field by means of an additional model that accounts for the effects of prevailing ocean characteristics on sound propagation. For this additional model, the MENCK group used analytical relations based on test data that were available for sound propagation as a function of distance accounting for water depth and seabed properties.



Underwater sound generation and propagation shown as a sequence of snapshots in time. Within a steel pile, the speed of sound is about 5,000 meters per second, while the speed of sound in water is about 1,500 meters per second — resulting in radiation patterns and specific inclination angle.

Numerical results from the simulation have been validated for the installation of the monopile on the FINO3, a government-sponsored wind energy research platform in the North Sea [4]. The comparison of measured and calculated sound pressure at a distance of 245 meters from the pile showed good correlation of the first pressure peak's amplitude. Beyond this initial analysis work, however, further validation is required: The peak sound pressure level observed near the pile is relatively high compared to the ambient underwater pressure, which might violate linear wave theory. A full FSI analysis to couple ANSYS Mechanical with ANSYS CFD fluid flow simulation software without the stated typical assumptions of linear acoustics may be applied for this purpose. ■

References

[1] Abromeit, C. *Licensing Requirements and Conditions in the German EEZ, Federal Maritime and Hydrographic Agency (BSH).* Proceedings of 24th Conference of the European Cetacean Society, Stralsund, Germany, March 2010.

[2] Nehls, G.; Betke, K.; Eckelmann, S.; Ros, M. *Assessment and Costs of Potential Engineering Solutions for the Mitigation of the Impacts of Underwater Noise Arising from the Construction of Offshore Windfarms;* BioConsult SH report on behalf of COWRIE Ltd.: Husum, Germany, September 2007.

[3] Moosrainer, M. Analyzing Vibration with Acoustic–Structural Coupling. *ANSYS Advantage,* **2009**, 3(2), pp. 40–42.

[4] Steinhagen, U. *Transient FE Simulation of the Hydro-Acoustics during Pile Driving.* Proceedings of ANSYS Conference & 27th CADFEM Users Meeting, Leipzig, Germany, November 2009.

Image courtesy Dag-Tor Afinse/StatoilHydro

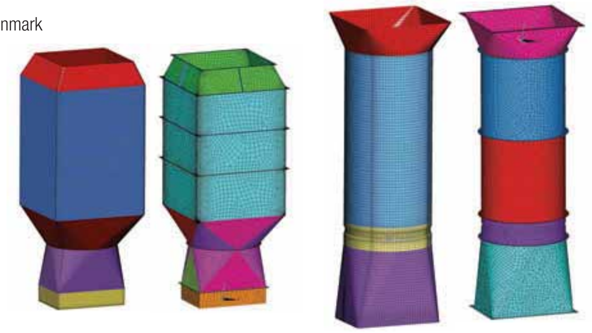
Tracking Down Vibrations Fast with **FSI**

With high-speed iterations between mechanical and fluids software, fluid structure interaction quickly pinpointed the cause of damaging vibrations and assessed new designs for offshore oil and gas equipment.

By Johan Gullman-Strand and Kenny Krogh Nielsen, Lloyd's Register ODS, Copenhagen, Denmark

Companies operating offshore oil and gas platforms can lose significant revenue for even a few days of downtime, so they must efficiently study and rectify equipment failures that could shut down any part of the operation. Case in point was the appearance of fatigue cracks and open tears in exhaust stacks for gas turbines, that powered electrical generators and natural gas compressors on a rig in the North Sea. The 10 meter-high welded sheet-metal structures safely direct the flow of 540-degree Celsius exhaust gases, with velocities up to 180 meters per second, up and away from the gas turbine. Field measurements with accelerometers placed on one of the stacks indicated extreme vibration levels at a frequency of approximately 20 Hz, particularly in the lower cone section where most of the cracks occurred.

The operator of the platform, StatoilHydro, engaged the services of engineering consulting firm Lloyd's Register ODS (LR ODS) to study the behavior of the stacks. The purpose of the project was to determine the root cause of vibrations in the existing design and to evaluate vibration levels of proposed new stack designs from two independent suppliers. The existing design was mostly rectangular for the various sections, but both new designs had cylindrical geometries in the mid-section of the stack. One of the new designs, D1, had fairly long plane wall sections.



Original Design

New Design D1

Engineers performed FSI iterations efficiently within a unified suite of software between models created with the same meshing tool: ANSYS ICEM CFD. Fluid models (left in each group) were constructed with volume elements, while shell elements were used for structural models (right).

Using engineering simulation, LR ODS studied the designs in greater detail than would have been possible through the time-consuming and expensive process of

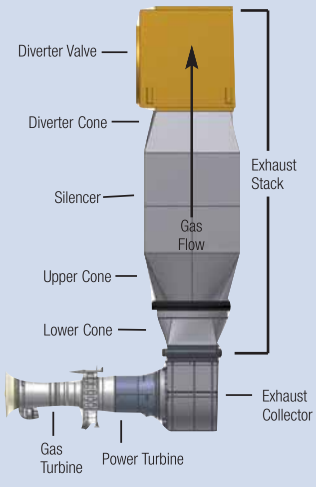
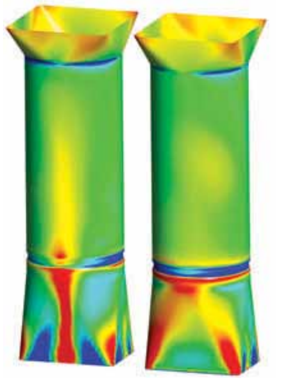


Diagram of exhaust duct

Anatomy of an Exhaust Stack

The gas turbine exhaust stack directs hot gases from the gas turbine and power turbine upward to the waste heat recovery unit (WHRU) or the bypass duct (dependent on diverter valve setting). The geometry of the stack is rather complex due to the arrangement of the gas turbine and existing ductwork as well as the need to minimize flow separation of the high-velocity gases.

An exhaust collector with a circular inlet and a rectangular outlet diverts gas flow 90 degrees from the gas turbine axis. In the original design, the collector was followed by a bellow that mechanically decoupled the part from the rest of the stack and allowed for thermal expansion of the duct structure. Next were two transition cones: a lower rectangular-to-circular cone and an upper cone that went back to a rectangular cross section. This double transition was based on historical reasons and not primarily designed for good flow quality. A long rectangular silencer mid-section followed, leading to a smaller-diameter diverter cone connecting the exhaust duct with a valve house. The majority of the fatigue cracks occurred in the lower cone, where FSI simulations were used to study resonant vibrations produced by turbulent flow of exhaust gases in the stack.



ANSYS CFX output from the FSI simulation displays averaged pressure coefficient distribution on the surface of design D1.

FSI stress levels computed by ANSYS Mechanical software for new D1 design at time 1.995 seconds, as seen from two different viewpoints

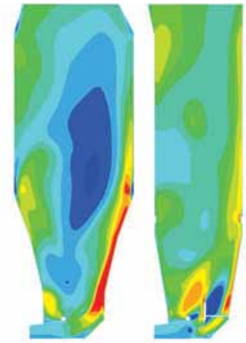
building and testing physical prototypes. First, engineers performed a modal analysis with ANSYS Mechanical software, which calculated relatively low first natural frequencies in the range of 15 Hz to 25 Hz. The mode shapes from this analysis indicated that the maximum vibration amplitude would occur in the lower cone of each structure.

Next, the engineering team performed a fluid structure interaction (FSI) simulation to study the turbulent flow of exhaust gases through the stack and the resulting pressure fluctuations on the sidewalls. By identifying pressure fluctuations from exhaust gas, the team could determine which vibration modes were excited. They could then calculate stress levels experienced by the vibrating structure. For this analysis, LR ODS used ANSYS CFX and ANSYS Mechanical software for fluids and structural computations, respectively.

For this project, the engineering team evaluated different methods for representing turbulent fluid flow, each well suited for particular applications. These included Reynolds averaged Navier–Stokes (RANS), large eddy simulation (LES), detached eddy simulation (DES) and unsteady RANS (URANS), a variant of the efficient RANS method in which flow can vary with time. Given the efficiency of URANS in handling time variations for relatively low-frequency excitations and LR ODS' experience with these various approaches, the company selected the URANS method for the exhaust stack FSI study.

The team performed a two-way iterative FSI coupled-field solution with a one-way limiter using the URANS model for ANSYS CFX calculations of flow pressures. These flow pressures were then fed into ANSYS Mechanical software for calculating the resulting structural stresses as well as sidewall deformations. To meet required solution accuracy, iterations proceeded in one-millisecond time increment steps — small enough to provide sufficient detail.

To simulate one second of stack operation, the software was set to perform 1,000 iterations with a run time of less than 12 hours — extremely fast compared to the six days sometimes needed for an FSI solution using DES models. Such high speed was possible mainly because iterations were completed so efficiently — all performed within a unified suite of software between models created with the same meshing tool: ANSYS ICEM CFD.



FSI analysis indicated the root cause of the vibrations for the original design — a large flow separation zone with low-frequency pressure fluctuations that coincided with very lightly damped duct wall natural frequencies. Simulation demonstrated that the recirculation zone was reduced significantly in the new D1 design.

FSI analysis indicated that the root cause of the vibrations for the original design was a large flow separation zone with pressure fluctuations occurring at the same frequency as the stack wall natural frequency. FSI results showed in a unique way how the large separated vortex structures caused the stack wall to vibrate severely. In these vibrations, cyclical stress levels exceeded material fatigue limits, and deformation amplitudes were greater than 2 millimeters.

Simulation demonstrated that the large recirculation zone seen in the original design was significantly reduced. Engineers modified the stack design in cooperation with the manufacturer, Mjørud AS, to increase the first structural natural frequencies beyond the dominant pressure fluctuation frequencies. The modified design also increased the structural damping using the thermal insulation. ANSYS meshing and simulation were instrumental in this design refinement, giving engineers insight into the vibrations and enabling the team to quickly evaluate the impact of various changes.

The manufacturer subsequently built and installed the new stack design on the oil and gas platform for the five gas turbine units. Field tests on one unit showed that the maximum single-frequency vibration amplitude level had been reduced by a factor of 30, and total vibration level was reduced by 80 percent. Since then, the five exhaust stacks have operated reliably for more than two and a half years. In addition to solving this complex FSI problem, the methodology developed by LR ODS has provided Mjørud with an efficient tool to quickly evaluate future stack designs, thus saving significant time and expense compared with troubleshooting problems in the field. ■

References

[1] Gullman-Strand, J.; Nielsen, K.K.; Hansen, L.V. DES and FSI for Industrial Applications, *Proceedings of the Fourth National Conference on Computational Mechanics*, pp. 7–24, **2007**, ISBN 978-82-519-2235-7,

[2] Nielsen, K.K. et al. Coupled CFD–FEM Analyses Used for Gas Turbine Exhaust Duct Vibration Troubleshooting, *Proceedings of Turbomachinery Symposium 2007*.

[3] Website: www.lr-ods.com.

Images courtesy StatoilHydro and Mjørud AS.

Image © iStockphoto.com/MichaelJay

Catch the Next Wave

Hydrodynamic simulation helps to deliver two- to three-times wave power efficiency improvement.

By Bradford S. Lamb, President, and Ken Rhinefrank, Vice President of Research and Development, Columbia Power Technologies, LLC, Corvalis, U.S.A.

If all the ocean's energy could be harnessed, it would produce more than 500 times the global energy consumption. The practical potential for wave energy worldwide is projected to be between 2 trillion and 4 trillion kilowatt hours per year. The World Energy Council estimates that about 10 percent of worldwide energy demand could realistically be met by harvesting ocean energy.

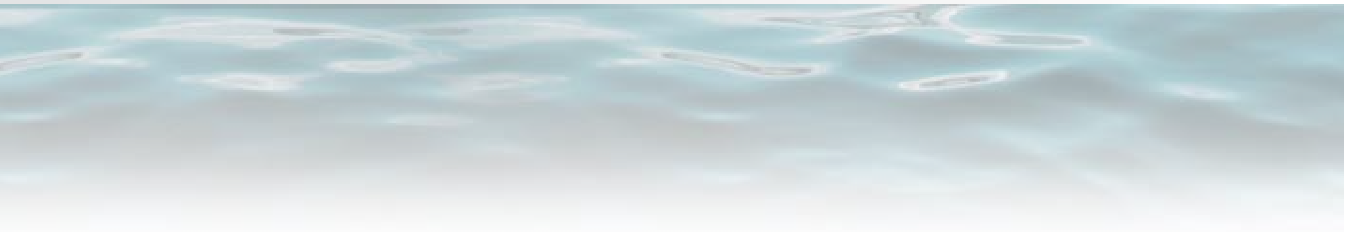
But wave power is a much less mature technology than solar or wind power or, especially, fossil fuel. A tremendous amount of work lies ahead in optimizing the design of wave power systems. Researchers must improve efficiency and reduce costs to the point that these systems can make a major contribution to meeting global energy requirements.

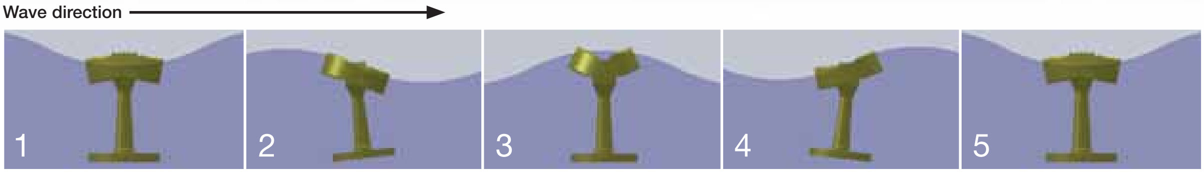
Columbia Power Technologies (COLUMBIA POWER), LLC, is attempting to harness this potential by developing commercially viable and scalable wave power generation systems. In conjunction with Oregon State University, the company is working to develop and commercialize innovative wave energy harvesting devices.

There are several key advantages of wave power:

- **Power density**: Wave power is much denser than other renewable energy systems, enabling wave parks to produce large amounts of power from a relatively small footprint.
- **Predictability**: The supply of energy from wave power can be accurately forecast several days in advance, enabling utilities to make precise sourcing plans.
- **Constancy**: Unlike solar power, which produces energy only when the sun is shining, ocean swells are available 24 hours per day.
- **Proximity to load centers**: Wave energy will not require substantial buildout of transmission capacity, since 37 percent of the world's population live within 60 miles of a shoreline, and 70 percent reside within 200 miles.

Preparing to test the wave power device





COLUMBIA POWER's wave power system: The wings and vertical spar react to the shape of the passing ocean swell. Each wing is coupled by a drive shaft to turn its own rotary generator.

The wave power industry, however, faces a major challenge since product developers have much less experience in the design of wave power devices relative to other renewable energy systems. Wave power companies need to rapidly advance efficiency and reduce costs of their designs to demonstrate viability to potential investors and customers. Other industries have taken decades or longer to develop technology to the point of commercial viability. But the wave power industry does not have that kind of time. To achieve its goals, it needs to rapidly improve designs while conserving limited capital.

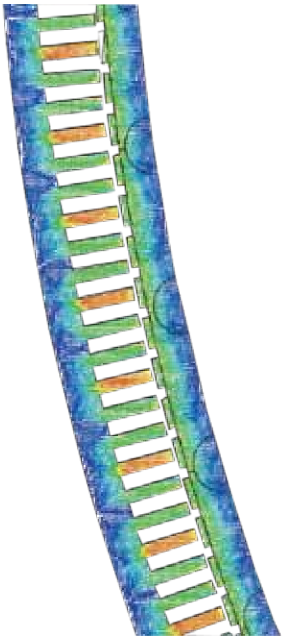
COLUMBIA POWER is focusing on development of direct-drive systems, which avoid the use of pneumatic and hydraulic conversion steps and their associated losses. The company believes that direct-drive systems are the future of wave power because they are more efficient and reliable as well as easier to maintain. The number-one design challenge was to optimize the design of the buoy to maximize the proportion of wave power transferred to the buoy. Relative capture width is a dimensionless measure of the efficiency of the device in capturing the available energy of the wave. A relative capture width of 1 means that the buoy has captured 100 percent of available wave energy.

As COLUMBIA POWER set out to determine the optimal shape for the buoy, engineers looked at five different hydrodynamic simulation software packages. The company selected ANSYS AQWA software because of its ease of use, and tests showed that it provided a better match with physical experiments than did competitive software. COLUMBIA POWER also valued that ANSYS AQWA offers both frequency and time domain solutions. Frequency domain solutions are faster, which makes them ideal for quickly evaluating a large number of shapes, while time domain solutions provide the high level of accuracy needed to refine to the best shapes in the later stages of the design process.

COLUMBIA POWER engineers developed an initial concept design in SolidWorks®, built a prototype and tested it at 1/33 scale in the Tsunami Wave Basin at the Hinsdale Wave Laboratory at Oregon State University. The team used high-resolution cameras to track light-emitting diodes on the buoy, measuring its motion in the waves. Engineers exported the concept design to ANSYS AQWA software and performed a time domain simulation while using a wave climate with the same amplitude and frequency as that measured in the wave tank. There was a very good match between the measurements and predictions from ANSYS AQWA. Since then, engineers have used ANSYS AQWA as their primary design tool to optimize the shape of the fiber-reinforced plastic (FRP) buoy.

COLUMBIA POWER engineers doubled efficiency of the buoy by using ANSYS AQWA to optimize its geometry.

COLUMBIA POWER has since evaluated over 350 different geometries with ANSYS AQWA in an effort to maximize the relative capture width of the buoy. At the same time, the company worked closely with Ershigs Inc., its structural partner that produces the FRP floats, to explore the manufacturability of various shapes and to ensure that the final design can be produced at a low cost. The company also looked at the survivability and environmental impact of proposed buoy designs. COLUMBIA POWER engineers used a sinusoidal wave shape and a suite of wave frequencies ranging from 2 seconds to 20 seconds for frequency domain simulations. The response amplitude operators calculated by ANSYS AQWA software were used in a post-processing routine written by COLUMBIA POWER engineers that calculates the relative torque and speed of the buoy as well as the relative capture width.

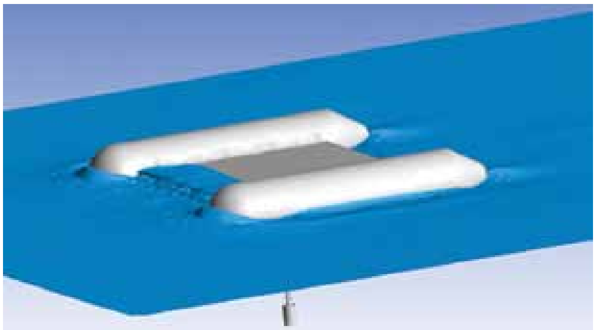


Maxwell software from ANSYS was used to optimize the generator design.

Once they felt that they were close to an optimal shape for the buoy, COLUMBIA POWER engineers moved to time domain modeling, which makes it possible to evaluate the nonlinear effects of the waves. The team evaluated the shapes that had proven best in frequency domain modeling against a variety of wave climates, including those found at seven different coastal locations around the world. At the same time, engineers began optimizing the power takeoff system that converts mechanical energy into electrical energy. ANSYS AQWA model results from frequency domain models were post-processed in Matlab® Simulink® to incorporate the power takeoff reaction torque and to compute power output. The ANSYS AQWA time domain models were coupled to a DLL that simulated both linear and nonlinear power takeoff operation. The DLL for the power takeoff model was developed in Matlab Real Time®. Engineers used the output from ANSYS AQWA to drive a numerical model developed in Simulink that simulates the power takeoff system and control strategy. The control strategy tunes the power takeoff to the wave climate by changing the amount of current produced by the generator, which, in turn, changes the mechanical load placed on the system. This makes it possible to consider in a single model the effects of different buoy shapes, power takeoff system designs and control strategies; it also helps to determine the power that would be generated by each approach in a variety of different wave climates.

COLUMBIA POWER recently began using Maxwell electromagnetic simulation software from ANSYS to optimize the design of the generator. Engineers evaluated three different electromagnetic simulation software packages and concluded that Maxwell was the easiest to use and the most stable. Maxwell is being used to analyze the electromagnetic performance of the generator while varying the air gaps between the rotor and stator, different magnet geometries, different magnet types, and different types of steel. The overall goal is to maximize the generator's energy output while minimizing its cost.

As a technology startup with far-from-unlimited funding, COLUMBIA POWER must be capital efficient. By focusing its development efforts on simulation and using physical testing judiciously as a verification tool, COLUMBIA POWER is moving forward in the development process much faster than would be possible using traditional development methods. ANSYS AQWA and Maxwell simulation software enable the company to make its mistakes in the computer, where they are far less expensive than in the ocean. ANSYS AQWA technology, in particular, helped to more than double the efficiency of COLUMBIA POWER's wave power system. COLUMBIA POWER has benefitted from the excellent technical support and productive training sessions provided by ANSYS. As a result, the company is on track to soon deploy the first ocean demonstration of its technology in Puget Sound. ■



Sensor float fluid–structure interaction (FSI) transient response to current flow

Sensor float bow nosing down due to flow on sensor below float, deck awash

Don’t Rock the Float!

Fluid–structure interaction allows designers to assess impact of waves on freshwater and offshore systems.

By Richard Grant, President, Grantec Engineering Consultants, Inc., Halifax, Canada

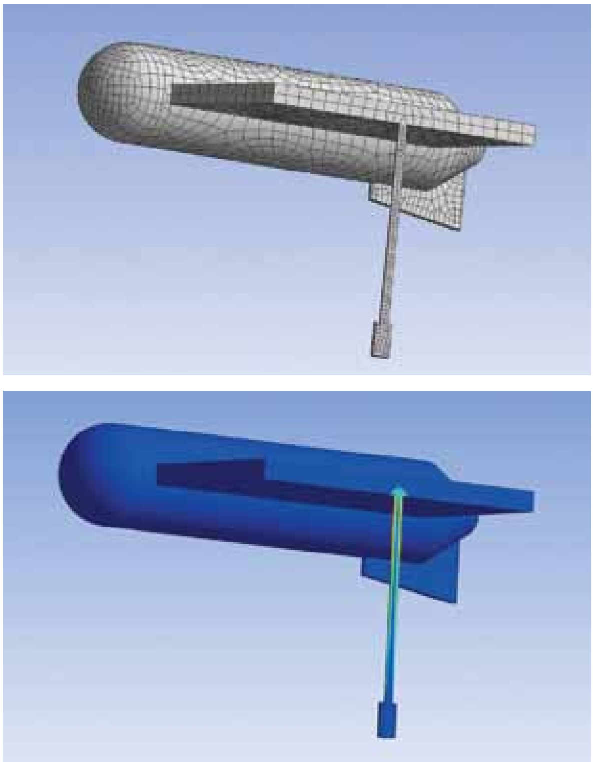
On a recent project commissioned by Environment Canada, Grantec Engineering Consultants, Inc. was tasked with developing a water quality monitoring float designed to carry a sensor for capturing environmental data. The float plays a role similar to — and looks somewhat like — a catamaran, though it is designed to be moored rather than driven by an engine or sails. The goal of the analysis was to minimize drag and ensure stability of the float as well as to develop specifications for the mooring system and structure. To meet this goal, Grantec used multiphysics simulation software from ANSYS to determine the fluid–structure interaction (FSI) by modeling the float and sensor under a wide range of water current and wave conditions.

Based in the maritime province of Nova Scotia on the east coast of Canada, Grantec and its engineers have an extensive background in both structural and fluids analysis helping customers in the defense, offshore, marine, manufacturing, energy and aquaculture fields advance new designs and systems. More recently, however, Grantec has often faced the challenge of how to combine these two analyses that have historically been performed separately. Previously, when the interaction between fluid and structure was critical, Grantec’s engineers needed to enter the results from the fluid dynamics software manually into the structural analysis software and vice versa. In contrast, ANSYS offers a solution integrating several of its most powerful and trusted fluids and structures simulation tools. With its multi-field solver, the ANSYS FSI solution provided Grantec’s team with a bidirectional capability for time-transient or steady-state analysis with moving or deforming geometry. Using ANSYS Multiphysics software, the Grantec engineers were thus able to evaluate both the structural part of the analysis and the fluid flow solution with just a single tool.

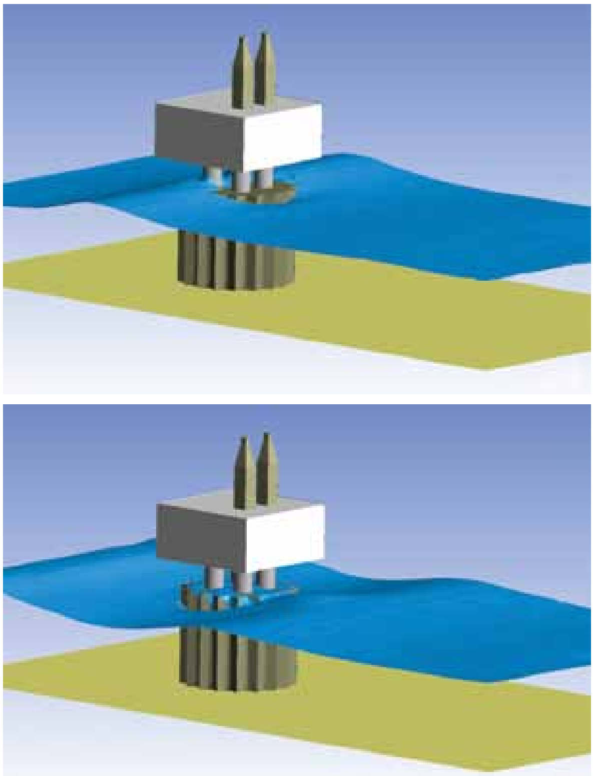
In the original float design, the team modeled the float and sensor as a flow obstruction, which accounted for the flow currents and wave loading on the float as well as buoyancy forces. They then evaluated the development of bow and stern waves that result from the resistance of the hull to fluid flow, just as with the hull of a ship. The software duplicated the vertical heaving and angular pitching of the float in response to different wave and current conditions. The impact forces from the waves calculated in the fluid simulation were automatically passed back to the structural model to more accurately simulate the stresses and deformations on the hull. Though they have little effect on fluid flow, the stresses are important because they make it possible to optimize the design of the hull to a much higher level than would be possible without them.

With the FSI solution from ANSYS, Grantec evaluated the performance of a wide range of hull profiles and mass distributions under different flow conditions, and it took advantage of parallel processing to accommodate larger models more efficiently than using a single-machine environment. In the initial series of designs studied, the sensor was fixed to the stern of the float and extended vertically into the water. The FSI results for these designs showed the force exerted by water currents on the sensor combined with the bow wave tended to push the bow of the float underwater in faster currents. It was not practical to solve this problem by simply changing the hull design, so the team tried a hinged connection between the sensor and the float to reduce the load transmitted from the sensor to the float. The hinged sensor, however, greatly increased the complexity of the simulation analysis.

Grantec addressed the new challenge of the hinged-sensor design by modeling the float with the sensor fixed in different hinge positions using the immersed pipe element in the structural portion of ANSYS Multiphysics software. Unlike the extensive approach used for the non-hinge designs, this new method provided a more simplified way to perform FSI analysis. With the immersed pipe element, the team applied wave and current loading to the structural model without the computational load involved in coupling it to a full fluid dynamics analysis. In the future, Grantec plans to use a moving mesh to perform a more complete FSI analysis including full fluid dynamics simulation that will evaluate the motion of the hinge in response to hydrodynamic forces.



Finite element mesh (top) and contours of stress (bottom) on a half model of the sensor float. The FSI analysis was performed to look at the effect of a fixed flexible boom on the float.



Waves washing over top of gravity-based structure of offshore platform (waves traveling to the right)

Beyond its studies of water quality monitoring floats, the company has done extensive work with engineering simulation to help create safer and more structurally sound offshore structures and systems. Grantec's engineers have also used the ANSYS Multphysics solution to assess gravity-based structures (GBSs) used to protect offshore oil drilling and production platforms from icebergs. GBSs rely on weight to secure them to the seabed, which eliminates the need for pilings in hard seabeds. Concrete GBSs are typically built with huge ballast tanks so they can be floated to the site and, once in position, sunk by filling the tanks with water. The Grantec team used FSI from ANSYS to simulate wave loading a GBS including the effects of massive waves from storms — also known as green water — coming over its top.

The company believes that its investment in ANSYS Multiphysics software has made a significant addition to its analytical capabilities. Clients seek out Grantec because of its track record in performing advanced engineering to solve very complex problems. ANSYS technology has helped put another tool in the Grantec toolbox that makes it easier to address design challenges that just a few years ago would have been much more difficult. ■

© iStockphoto.com/Alex Slobodkin

Getting Connected

ANSYS EKM helps engineering teams improve productivity by organizing simulation data locally and worldwide.

By Michael Engelman, Vice President, Business Development, ANSYS, Inc.

Connections are important, whether involving family, friends — or simulation data. New technologies hold great promise for expanding our reach into new networks, but they must be managed correctly to yield expected gains.

Successful companies recognize that simulation expertise is a valuable commodity, and that sharing this information is vital to streamline the development process and bring innovative products to market faster — a key to corporate success. Often, this expertise is localized to individuals or workgroups that informally function as mini centers of excellence. These workgroups tend to develop and maintain a company's best practices for simulation, as well as warehouse the simulation results from the projects they work on. As an engineering organization becomes more complex, projects begin to involve engineers with specialized expertise who often reside in different locations. No matter what continent they work from, engineers must collaborate and get connected on projects as if they shared a common office. More and more, companies are turning to systems like ANSYS Engineering Knowledge Manager (EKM) to capture and share simulation knowledge.

Consider the example of MANN+HUMMEL, one of the world's major automotive suppliers. Martin Lehmann, head of the organization's Simulation Filter Elements, commented, "We have engineers in Europe and India who frequently need to share models, CAE data and simulation results. They also need to collaborate in real time while performing CAE analysis. ANSYS EKM allows us to transfer and share simulation data very effectively. The product's extensive data management capabilities make it straightforward for our engineers to organize and track multiple versions of files that are created during a typical design and analysis cycle."

While workgroups have a common need to stay connected, the needs of each simulation expert are unique. Any system designed to capture and reuse simulation data and best practices must make individuals' jobs easier and improve their productivity, or it will have poor adoption rates.

"First of all, a simulation data management (SDM) system must meet the needs of its primary users, CAE analysts and workgroups," said Keith Meintjes, research director for CAE at Collaborative Product Development Associates, an organization that provides companies with objective information for assessing technology, business goals, and objectives for engineering and manufacturing. "SDM is not simply an extension of a PDM system or a PLM strategy; CAE is far too complex for top-down solutions. Companies should first concentrate on implementing SDM for individual and workgroup productivity and for simulation quality. Once that is in place, they can consider aggregating the data and metrics to gain enterprise-wide benefits. SDM should be seen as a strategy to capture and reuse engineering knowledge and intellectual property. The benefits will accrue from dramatically improved engineering processes, not from populating some PDM database."

ANSYS EKM manages the richness and complexity of simulation data in a way that makes it easy for the individual to function in a workgroup, while making it possible to share the context of simulation with CAE users enterprise-wide. This software system allows organizations to get connected to streamline processes, protect intellectual property, share best practices and foster innovation. ■

CHURNING WIND INTO POWER

By Ulrich Bock, Service Engineer
CADFEM GmbH, Grafing, Germany

Systems engineering can aid in designing an efficient wind turbine generator.

A wind turbine generator consists of all the components required for mechanical-to-electrical energy conversion. The turbine blades, generator and electrical converter are the three main functional assemblies for a wind-energy generator.

The most important component within the interacting elements of a wind turbine system is the generator that creates electric energy. The output of the generator supplies power to the converter subsystem, which conditions this electrical power. The power is then transferred onto the grid.

Some of the electrical components used in this conditioning and transfer process include transformers, capacitors, inductors and power electronics, as well as cables and bus bars.

Engineers who design wind turbines face a significant challenge: to accurately determine specifications of various individual components — which often originate from different manufacturers — and then connect them into an efficient system. ANSYS software provides simulation tools that apply to all the sections of a wind turbine generator, leading to an efficiently designed and optimized system as a whole.

SYSTEM SIMULATION

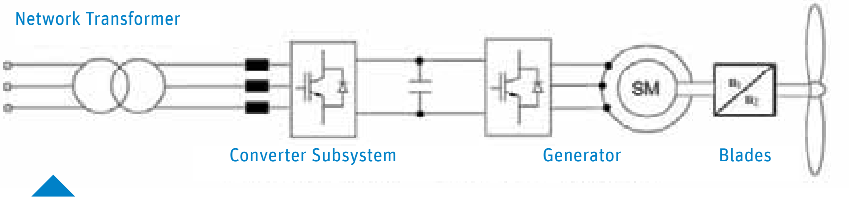
Without using simulation tools, it is almost impossible to determine the performance of a complex heterogeneous system, such as a wind turbine, that incorporates the high efficiency requirements needed for the electrical system.

The heart of the ANSYS solution methodology for electromechanical systems is ANSYS Simplorer software. Using this technology, the engineer can set up a complex system entirely using analytical models. By employing detailed finite element method (FEM) models and a broad set of ANSYS simulation tools, users have the capability to analyze, optimize and embed all components in the overall end-to-end system.

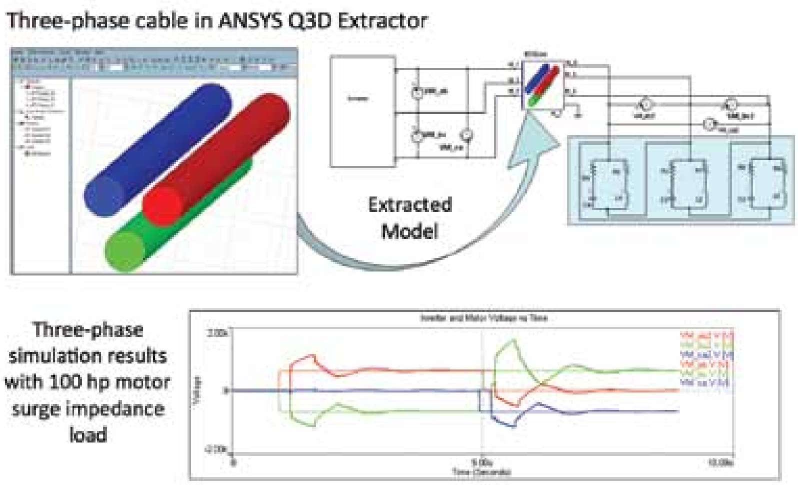
Many design requirements for these electrical systems are complex and can be determined only via simulation. These include network coupling design and integration of power control based on existing electrical generator characteristics as well as admissible power fluctuations and/or suppression of harmonics.

Employing simulation tools can be useful for testing critical operational conditions such as a short circuit that might occur in close proximity to (or at a specific distance from) the generator, admissible thermal loads, or the design of networks to protect against other effects, such as electrical surges. Other ANSYS software can be used in conjunction with Simplorer:

- **ANSYS RMxprt, a tool specific to design of rotating electrical machines, for modeling the generator, making initial sizing and performance decisions, and preparing the model for simulation using Maxwell**
- **ANSYS Maxwell 2-D and 3-D tools for finite element simulation of the generator**
- **ANSYS Q3D Extractor for analysis of parasitic influences in and between cables and/or bus bars**
- **ANSYS Mechanical for structural analysis**
- **ANSYS CFD for determining wind speed/ turbine speed relationship**



Electrical system of wind energy plant



Simulation of three-phase cable in ANSYS Q3D Extractor embedded in ANSYS Simplorer

ANALYSIS OF SYSTEM COMPONENTS

The capability to analyze systems and components is seamlessly integrated within the ANSYS family of simulation tools. For example, you can analyze the thermal load for, or impact upon, the different elements of the electrical network by employing Maxwell 3-D in combination with ANSYS Mechanical software. In the same way, you can determine the feedback for temperature-dependent parameters via Maxwell and simulate the resulting deformations with ANSYS structural mechanics, or conduct further thermal studies with fluid dynamics tools. As another example, electrical forces between the conductive rails and any resulting deformations can be determined by combining Maxwell 3-D with ANSYS structural mechanics products.

WIND ENERGY APPLICATIONS

Generators

Wind power generation employs various types of generators, such as synchronous generators and doubly fed induction machines. Engineers can analyze and optimize device efficiency with ANSYS tools. Initial design specifications can be entered into RMxprt along with the geometric and material data to obtain a preliminary design, and then a parameterized equivalent circuit diagram can be exported to Simplorer. Alternatively, using information from RMxprt, you can automatically create the generator model in Maxwell 3-D or 2-D and directly link to Simplorer. Using this unique coupling allows you to consider eddy currents in the generator created by switching operations in the converter.

Converters

Both conversion from three-phase current to direct current (rectifier) and from direct current to three-phase current at a defined frequency (power inverter) are relevant in wind turbine design. To enable full design and analysis capabilities, conventional insulated gate bipolar transistor (IGBT) power electronic semiconductor switches, gate turn-off thyristors (GTOs) and thyristors are available in Simplorer in different levels of detail.

In simple terms, these semiconductor devices represent control switches. Depending upon the level of accuracy needed when investigating the device switching behavior, you can apply Simplorer models of varying details as well as semiconductor valve SPICE models. To allow customization, IGBT parameterization tools allow the appropriate model to be generated and simulated based on the data sheet of the specific type of IGBT.

Transformers

Simplorer enables simulation of different transformer models as equivalent circuit diagrams. Coupling with Maxwell 2-D and 3-D for more detailed testing is available.

Cables and Bus Bars

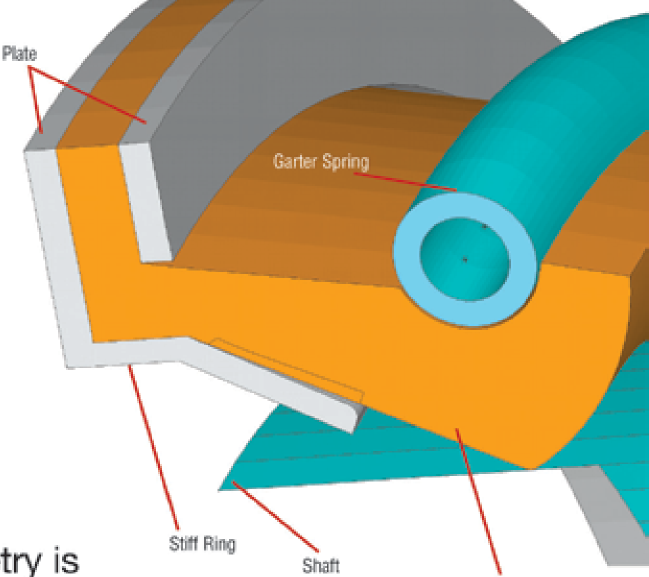
With the Q3D Extractor tool, you can quickly generate a parameterized equivalent circuit diagram from the 3-D cable geometry, which can then be exported to Simplorer, where it retains its connectivity. This makes it possible to analyze traveling waves, surge voltage and associated parasitic effects.

Controllers

Simplorer depicts controllers directly on the simulation sheet and couples these controllers with the electrical switches.

ANSYS Simplorer, when used as a system simulator in conjunction with the RMxprt, Maxwell, Q3D Extractor, ANSYS Mechanical and fluid dynamics tools, provides an efficient platform for simulating an entire wind turbine electromechanical system. Using systems simulation, wind turbine designers can meet overall requirements as well as specific requirements for components. Finally, R&D teams can integrate structural simulations for robust systems design.

This article was originally published in German in Infoplaner *magazine from ANSYS channel partner CADFEM.*



Radial lip seal geometry simulation model

Predicting Wear in Radial Seals

Finite element analysis is performed in a step-wise approach in which seal geometry is re-meshed with each load cycle to account for wear-off of material at the contact surface.

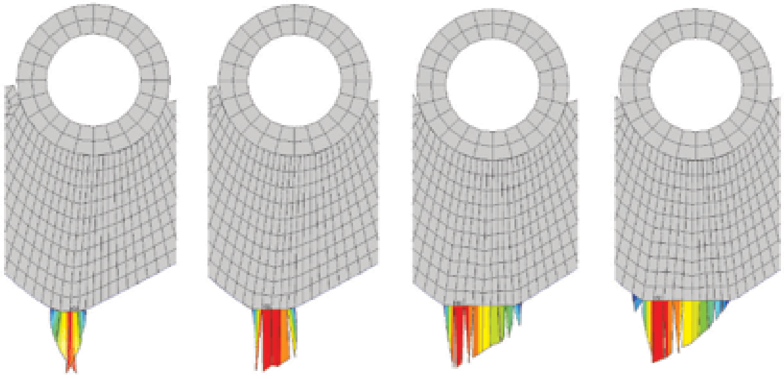
By Zhichao Wang, Manager, Analytical Services, Emerson Climate Technology, Ohio, U.S.A.

Radial shaft seals (including lip seals) made of elastomers or low-friction polytetrafluoroethylene (PTFE) materials are used in a wide range of products, including aircraft, vehicles and industrial equipment for sealing rotating shafts — primarily to keep out contaminants and keep in lubricating oil. A garter spring typically is used to create an adequate initial force between the shaft and the seal before high working pressure is built up. The seal contact pressure under the working pressure is a critical factor in seal performance and wear. This contact pressure is extremely difficult to measure because of the complexity of seal configuration, the size of contact area, and continuous changes in the contact profile due to material being worn off over the life of the seal.

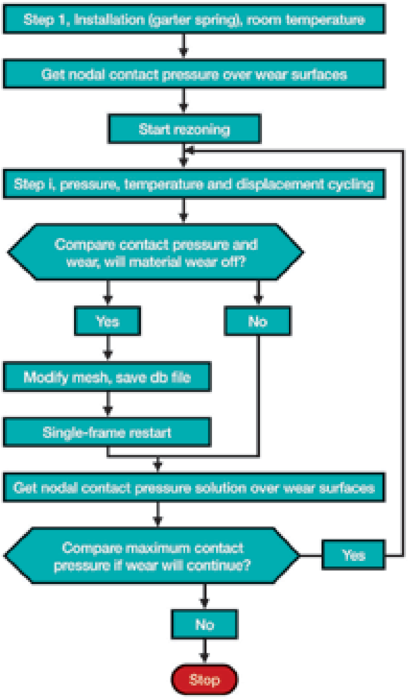
At Emerson Climate Technology in Ohio, U.S.A., ANSYS Mechanical software is used extensively as a powerful tool to gain a more thorough understanding of seal deformation and contact pressure. To perform a realistic simulation and obtain accurate results, the analysis is performed in a step-wise approach in which the seal geometry is remeshed with each loading cycle to account for the effect of material wear at the contact surface. The simulation is performed using the single-frame restart feature and a non-standard re-meshing procedure for each solution cycle.

The PTFE material is temperature-dependent, time-dependent and pressure-sensitive. Restart preserves the stress and strain history for each cycle, moving nodes using solutions of the previous step and saving the modified geometry into a database file from step to step. Hence, the mesh of the wear zone is modified continuously as a function of contact pressure and sliding velocity.

The process accurately represents material removed from cycle to cycle as a smooth function of contact pressure, in which contact pressure gradually evolves with the progress of material worn off for the number of cycles. Results clearly indicate that both the distribution and contact pressure of the seal change continuously due to the loss of material. Since contact behavior strongly impacts lip seal performance, gaining this insight has been a key to optimizing seal design and improving product quality and reliability at Emerson Climate Technology. ■



Evolution of seal wear from cycle to cycle: The change in the interface at the bottom of the seal area (lower grey edge) illustrates a change in shape; contact pressure is plotted across this lower edge, where the red areas indicate regions of maximum pressures of 19, 9.5, 3.4 and 1.6 MPa, from left to right respectively.



Simulation flow chart for single-frame restart procedure used for FEA modeling of a continuously wearing seal geometry

Separating the Streams

Multiphase simulation can improve performance of oil and gas separation equipment.

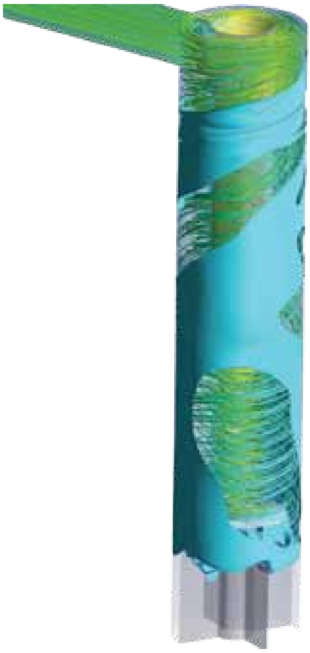
By David Stanbridge, Managing Director, Swift Technology Group, Norwich, U.K.

Separators are used throughout the oil and gas industry to split production fluids into components of oil, gas and water (as well as contaminants). On an offshore facility, the equipment is found in many parts of the overall process. The initial separator, usually referred to as first-stage, separates the initial stream into distinct gas, oil and water streams. These streams are then individually processed. Poor separation performance can hinder overall production; in some cases, platforms produce only 50 percent of design capacity due to poor separation.

The industry has used computational fluid dynamics (CFD) extensively to troubleshoot separation equipment performance with different methodologies. Most common is segregated single-phase simulation, in which gas and liquid phases are analyzed separately. Multiphase volume of fluids (VOF) simulations are useful in analyzing liquid sloshing behavior in separators secured to moving platforms. This sloshing analysis is usually carried out in combination with a user-defined function that adjusts gravity and applies three inertial forces: Coriolis, Euler and centrifugal. Historically, fluids neither enter nor leave the vessel.

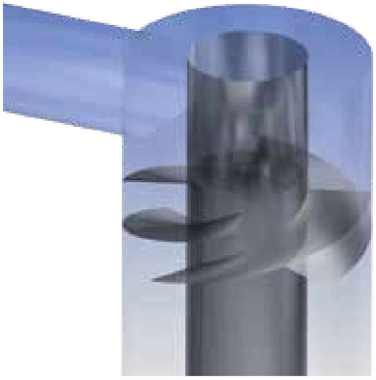
Benefits of Multiphase Simulation

As new separation equipment becomes smaller and flow rates exceed the design capacity of existing equipment, end users are questioning the accuracy of both the segregated single-phase approach and VOF for sloshing. Extended use of multiphase simulation is now possible as a result of enhancements to computer power and ANSYS FLUENT capabilities. Software improvements have led to reduced run times; multiphase and turbulence models have a greater ability to handle primary and secondary phases. The multiphase method overcomes the limitations of segregated single-phase and VOF approaches. It also allows for detailed analysis of interphase interactions, providing more realistic results. Swift Technology Group has studied two types of separation devices that use the multiphase method. The company is a technology-driven organization that offers complete end-to-end product development for the aircraft, marine, automotive, oil and gas, and renewable energies industries.



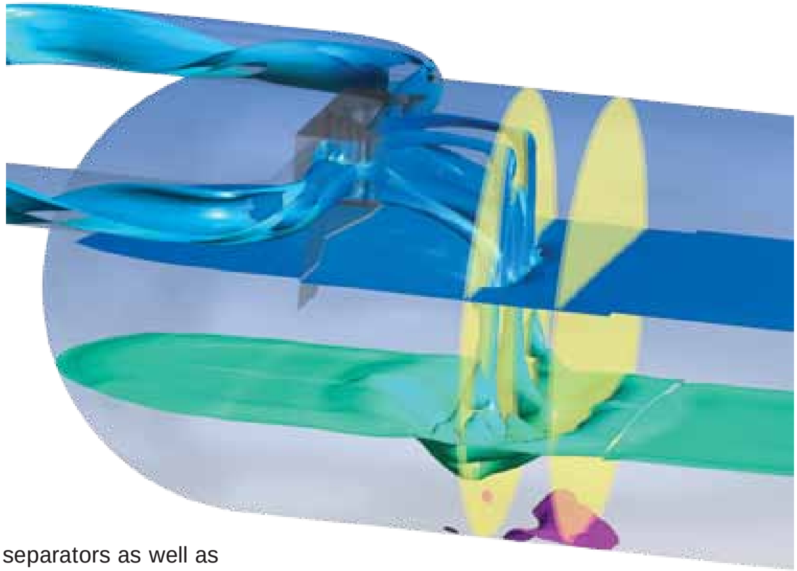
Simulation of a vertical cyclone vessel designed to remove bulk liquid from the feed stream. Pathlines of primary gas phase show where the liquid has a concentration of more than 25 percent.

Droplet separation is fundamental to good separation. The most common equipment for droplet separation is vertical or horizontal vessels that use gravity as the driving force. More-compact separation equipment often uses cyclones. By spinning the flow, employing a standard tangential inlet, or using more-elaborate swirl elements, cyclones can generate accelerations many times that of gravity to potentially provide more efficient separation in a smaller amount of space. However, many other considerations must be investigated. Traditionally, cyclonic equipment required exhaustive prototyping and testing to ensure that the many negative consequences were designed out of the final product — a lengthy and costly exercise. In a recent R&D program for cyclone development, Swift researchers found that the time for each design change cycle was approximately eight weeks at a cost of around £45,000 (approximately $73,000 U.S.) per cycle, with



Swirl element fitted in the cyclone

seven changes required. By using CFD, each change can be modeled in two weeks, requiring only one actual test — saving a total of more than £300,000 (approximately $485,000 U.S.). Note, however, that it is difficult to quantify the exact benefits of simulation in every case.



Multiphase simulation within a horizontal three-phase separator with inlet piping, a vane-type inlet device and full-diameter perforated baffles. The lower layer of fluid is water; above that is the oil phase with the inlet device in the gas phase of the vessel. The pink area at the bottom of the vessel shows where sand entrained in the water phase will initially settle.

Simulation of Separation Equipment

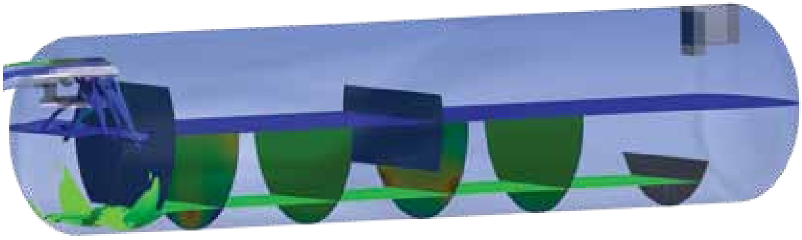
There are many examples in which the mixture multiphase model has been used to analyze separation within cyclonic equipment. The model is applicable for dilute-to-moderately dense volume loading, for low-to-moderate particulate loading, and for cases in which the Stokes number is less than 1. The simplified model can be used for hydrocyclones — equipment whose main function is to separate final oil droplets from water prior to disposal at sea. The comprehensive Eulerian multiphase model is applicable to the complex flows that are found in the most common types of separation equipment designed to remove bulk phases as well as re-entrained droplets. Users can enhance their analysis of cyclonic flows by applying the Reynolds stress turbulence model without limitation for all primary and secondary phases.

One important part of separator analysis is commonly overlooked: the impact of upstream piping. This system has a large effect on the distribution of fluids within the vessel. The simulation examples provided — horizontal and vertical gravity-driven separators as well as cyclone-based separators — incorporate the impact of upstream piping.

It is difficult to accurately validate the simulation results of the installed vertical cyclone and separator. Simulation has been shown to accurately capture both flow field and separation performance in lab and pilot test rigs. [See references.] Using these modeling strategies as well as exhaustive testing performed over many years, all the critical aspects of the flow are correctly resolved and indicate the key performance characteristics. As a result, Swift is changing out the internal components of many vessels based upon simulation results.

The main function of a horizontal three-phase separator is to split a feed stream into discrete gas, oil and water streams. Normally, gas is the primary phase, and the two liquid phases are secondary. These liquid phases form droplets that are entrained in the gas phase, and they produce a film on the pipe walls leading to the separator. The first component in the separator is the inlet device, whose primary function is to provide a coarse separation of gas and liquid phases. The gas phase continues along the top of the vessel, while the liquids drop to the bottom of the separator. At the bottom of the vessel, the two liquid phases separate, with the water at the bottom and the oil forming a layer between the water and gas phases.

In most cases, perforated baffles are used along the length of the horizontal vessel to control liquid phase flows and to distribute them evenly across the available cross-sectional area of the vessel, minimizing axial velocity and maximizing separation. The Eulerian model is required in this



The complete length of a typical horizontal separator: The blue layer represents the interface between gas and oil phases, and the green layer represents the interface between oil and water. The vertical blue areas represent part diameter perforated baffles. Along the length of the vessel, four contours show velocity distribution in both oil and water phases.

Analysis of a vertical production separator with a vane-type inlet device shows that the inlet pipe keeps much of the liquid on one side of the vessel — leading to non-optimal separation.

type of simulation because of the number of fluid regime changes.

In a vertical production separator with a vane-type inlet device example, gas and liquid are introduced at the start of the pipe run to the separator vessel. The pipe routing causes the liquid to be biased to one side of the vessel — which does not produce optimal separation and, in some cases, can lead to the gross carryover of liquid though the vessel's gas outlet.

In conclusion, Swift researchers have found that ANSYS FLUENT software can model — to a high degree of accuracy — many combinations and permutations of separators available within the industry. ■

References

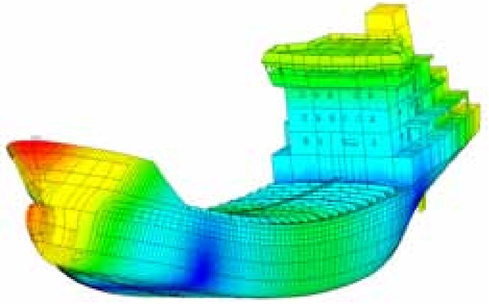
Slack, M.D.; Prasad, R.O.; Bakker, A.; Boysan, F. Advances in Cyclone Modelling Using Unstructured Grids. *Trans. I. Chem. E.*, **2000**, 78(A), pp. 1098–1104.

Cokljat, D.; Slack, M.; Vasquez, S. *Reynolds-Stress Model for Eulerian Multiphase*. 4th Symposium on Turbulence, Heat and Mass Transfer, **2003**, Antalya, Turkey, pp. 8.

Slack, M.D. *Best Practice Advice for AC3-03 Cyclone Separator, Thematic Network for Quality and Trust in the Industrial Application of CFD* (qnet-cfd). Thematic area 3, **2003**, Barcelona, pp. 5. http//www.qnet-cfd.net

---

TRANSPORT



7,500-dwt chemical tanker first bending vibration mode

Designing Safe and Reliable Ships

Marine engineers use upfront simulation to reduce vibration.

Ocean-going merchant ships are continually slammed, not just by shifting natural forces but also by loads, such as forces generated by the propellers and engine. These forces have the potential to generate vibrations that can make life miserable for the crew and, over time, damage the ship.

Delta Marine Engineering Co., an engineering and consultancy services provider for the design of various types of ships including oil tankers, must design these vessels to keep structural vibrations to low levels. This design challenge requires a solid understanding of the structural behavior of the ship and its interaction with the surrounding water.

Complex fluid and structural interactions govern the performance of a ship, so Delta Marine uses ANSYS software early in the design process to understand the intricacies. Engineers can identify vibration and other problems and make changes, such as altering a propeller design, to get the design right the first time. This avoids the need for expensive changes that could run into millions of dollars if the problem is not discovered until after the ship is launched.

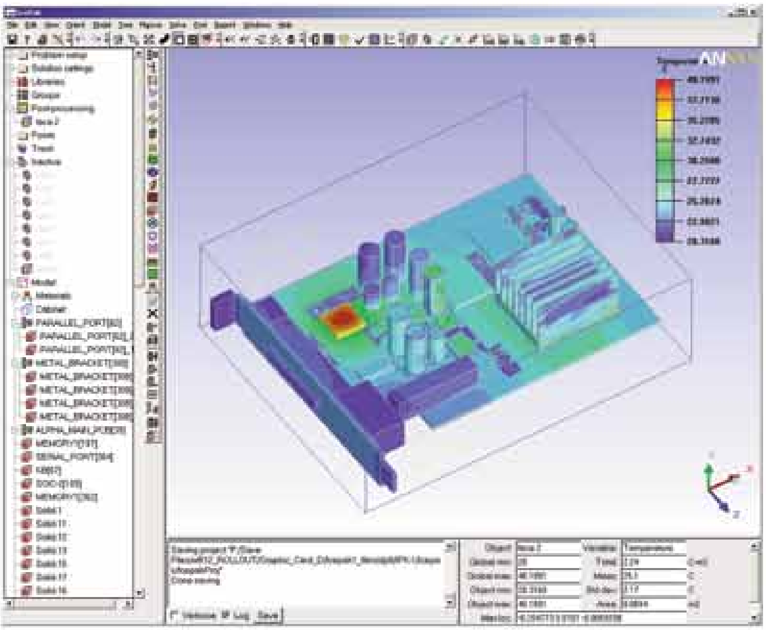
Each ship presents unique engineering challenges, so the company performs advanced engineering analyses for every single new design it creates, which enables Delta Marine to build safe and reliable designs. Using engineering simulation makes it possible to evaluate many more designs, resulting in substantial performance improvements. ■

Staying Cool with ANSYS Icepak

Thermal management solution predicts air flow and heat transfer in electronic designs so engineers can protect heat-sensitive components.

By Stephen Scampoli, Lead Product Manager, ANSYS, Inc.

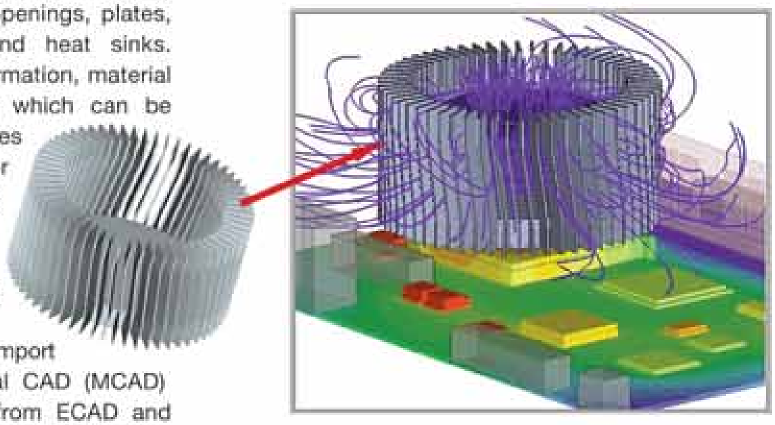
ANSYS Icepak technology is aimed at one of the most significant challenges facing engineers designing electronic assemblies: dissipating thermal energy from electronic components to prevent premature component failure due to overheating. This fully interactive software is used to evaluate the thermal management of electronic systems in a wide range of applications, including simulation of air flow in enclosures, analysis of temperature distributions in chip and board-level packages, and detailed thermal modeling of complex systems such as telecommunications equipment and consumer electronics. By predicting air flow and heat transfer at the component, board or system level, the software improves design performance, reduces the need for physical prototypes and shortens time to market in the highly competitive electronics industry.



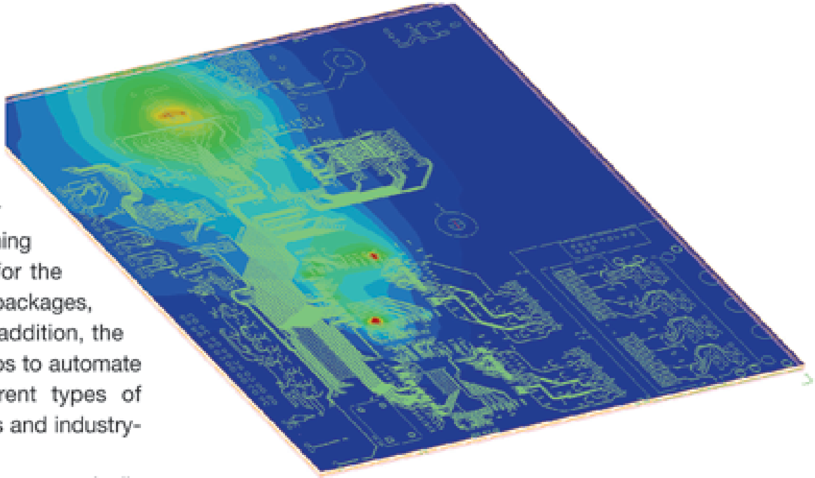
ANSYS Icepak software predicts the temperature profile in a computer graphics card.

Based on powerful computational fluid dynamics (CFD) simulation, ANSYS Icepak technology has a specialized user interface that speaks the language of electronics design engineers. Models are created by simply dragging and dropping icons of familiar predefined elements including cabinets, fans, circuit boards, racks, vents, openings, plates, walls, ducts, heat sources, resistances and heat sinks. These "smart objects" capture geometric information, material properties and boundary conditions—all of which can be fully parametric so a user can easily enter values to precisely match application requirements or to study what-if scenarios. The software also includes extensive libraries for standard materials, packages and electronic components such as fans — including fan geometry and operating curves.

As a further modeling aid, the software can import both electronic CAD (ECAD) and mechanical CAD (MCAD) data from a variety of sources. Geometry from ECAD and MCAD data sources can be combined with smart objects to quickly and efficiently create models of electronic assemblies. For



Simulation results of a fan-cooled processor heat sink attached to a printed circuit board



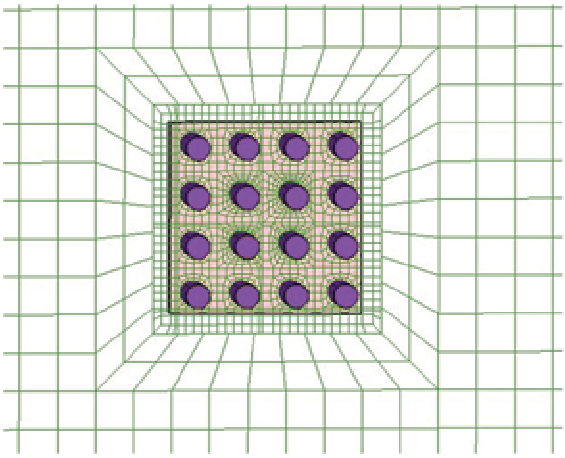
Thermal simulation of a PCB using the power map data imported from SIwave

instance, a system model of a computer enclosure could easily be generated by combining MCAD data for the enclosure, ECAD data for the printed circuit boards (PCBs) and electronic packages, and smart objects for other components. In addition, the ANSYS Icepak solution includes many macros to automate the creation of geometry, including different types of packages, heat sinks, thermo-electric coolers and industry-standard test configurations.

Another productivity feature is the ability to automatically generate highly accurate body-conformal meshes that represent the true shape of components rather than a rough stair-step approximation. Meshing algorithms can generate both multi-block and unstructured hex-dominant meshes. Algorithms also distribute the mesh appropriately to resolve the fluid boundary layer. While the meshing process is fully automated, users can customize the meshing parameters to refine the mesh and optimize the trade-off between computational cost and solution accuracy. By grouping objects into assemblies, the mesh count can be further optimized by meshing each assembly separately and automatically combining them before running the solution. This meshing flexibility results in the fastest solution times possible without compromising accuracy.

ANSYS Icepak uses the state-of-the-art ANSYS FLUENT computational fluid dynamics solver for the thermal and fluid flow calculations. The CFD solver solves the fluid flow and includes all modes of heat transfer — conduction, convection and radiation — for both steady-state and transient thermal-flow simulations. The solver also provides complete mesh flexibility, and this allows the user to solve even the most complex electronic assemblies using unstructured meshes, providing robust and extremely fast solution times.

Once the solution is complete, ANSYS Icepak software provides a number of different methods for visualizing and interpreting results. Visualization of velocity vectors, temperature contours, fluid particle traces, iso-surfaces, cut-planes and two-dimensional XY plots of results data are all available. The software also offers customized reports that allow users to identify trends in the simulation along with the ability to report fan and blower operating points. Reports including images can be created in HTML format for distributing the results data.



Mesh around a pin fin heat sink follows the geometry of the part without any approximation.

ANSYS Icepak tools can interface with other products in the software portfolio from ANSYS to allow comprehensive multiphysics simulation of electronic components. One option is the ability to import a power distribution map from SIwave; this simulation software from Ansoft extracts frequency-dependent electronic circuit models of signal and power distribution networks from device layout databases for modeling integrated circuit packages and printed circuit boards. Based on the results from an SIwave simulation, users can import the DC power distribution profile of printed circuit board layers into ANSYS Icepak software for a thermal analysis of the board. The coupling between the two packages allows users to predict both internal temperatures and accurate component junction temperatures for printed circuit boards and packages.

ANSYS Icepak software can export temperature data from a thermal simulation to a structural mechanics model to calculate thermal stresses of electronic components. With the demands of today's high-performance electronic devices, electronic components are becoming more complex and using more exotic materials. These newer materials have widely varying thermal and mechanical properties and are being subjected to higher temperatures during both manufacturing and usage. These varying material properties and temperatures can result in significant thermal stresses, which can bring about fatigue-based failure of the components. ANSYS Icepak software, together with ANSYS Mechanical technology, allows users to evaluate both the thermal and mechanical aspects of the design.

ANSYS Icepak technology in conjunction with SIwave and ANSYS Mechanical products provides a full portfolio of software to meet the simulation requirements of the electronics design engineer. ANSYS continues to be a leader in providing solutions to the electronics industry — solutions that provide the high-fidelity electrical, thermal and structural simulations required to meet the challenges of today's product development demands. ■

Every product is a promise

For all its sophisticated attributes, today's modern product is, at its core, a promise.

A promise that it will perform properly, not fail unexpectedly, and maybe even exceed the expectations of its designers and users. ANSYS helps power these promises with the most robust, accurate and flexible simulation platform available.

To help you see every possibility and keep every promise.

Realize Your Product Promise™

ANSYS®

To learn more about how leading companies are leveraging simulation as a competitive advantage, visit **www.ansys.com/promise**

ANSYS, Inc.
www.ansys.com
ansysinfo@ansys.com
866.267.9724

ANSYS is dedicated exclusively to developing engineering simulation software that fosters rapid and innovative product design. Our technology enables you to predict with confidence that your product will thrive in the real world. For more than 40 years, customers in the most demanding markets have trusted our solutions to help ensure the integrity of their products and drive business success through innovation.

© 2012 ANSYS, Inc. All Rights Reserved.

ANSYS, Inc. does not guarantee or warrant accuracy or completeness of the material contained in this publication. ANSYS, Aqwa, CFX , DesignXplorer, EKM, Engineering Knowledge Manager, Fluent, Full-Wave SPICE, HFSS, ICEM CFD, Icepak, Maxwell, Mechanical, Meshing, Multiphysics, Nexxim, Polyflow, Professional, Q3D Extractor, RMxprt, SIwave, Simplorer, Structural, Workbench, Ansoft Designer, Realize Your Product Promise, Simulation-Driven Product Development, and any and all ANSYS, Inc., brand, product, service, and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc., or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other brand, product, service, and feature names or trademarks are the property of their respective owners.