ANSYS

ADVANTAGE™

Excellence in Engineering Simulation

2014

SPECIAL ISSUE: Oil and Gas

PRODUCT RELIABILITY AND PERFORMANCE

Accelerate product reliability and performance with systematic use of engineering simulation.

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

The oil and gas industry's sharp focus on safety and reliability is based on the economics of oil and gas production, maintenance and unscheduled downtime — and along with that comes environmental stewardship and protecting human life. As project costs and complexity increase, companies develop continuous-improvement processes to reduce equipment and product failure to increase operational reliability at drilling, production and processing sites. This industry strives for zero accidents and, therefore, undertakes extraordinary measures to avoid loss to human life, environment and capital. Equipment reliability is the key to drive operational excellence and profitability, reduce resource waste, eliminate unnecessary downtime, decrease over-design and unplanned maintenance as well as non-productive time, and create smart intervention and prevention strategies. In fact, the industry spends a large part of its R&D budget on developing reliable products. The challenge is to develop these products for real-life conditions, environments that are often impossible to replicate with experiment.

Many variables must be considered when designing a new product. Because it isn't possible to test and prototype every permutation, some designers over-design parts or focus on the 20 percent of equipment and processes that they consider the main source of a problem. For existing equipment and facilities in service, some rely heavily on inspection and data gathering, incorporating historical data. These practices miss the mark: They can lead to poor product design and may cost millions of dollars for field testing along with related maintenance and inspection expenses. Furthermore, these procedures provide little insight that can be applied systematically and/or scaled to developing novel concepts and ensuring product performance during off-design conditions.

To guarantee product performance and compliance with industry standards and regulations, engineers must apply reliability practices to a wide range of equipment, products and processes — for example:

- Pumps
- Actuators
- Sensors
- Valves
- Transmitters
- Drill systems
- Pressure and flow control devices
- Controlling software
- Electronic and electrical devices
- Wireless signals

TABLE OF CONTENTS

Independent of how data is gathered (probability, historical data, etc.) or what analysis method is employed, the business impact of product failure is often much more than the initial cost of getting the design right, not to mention the price of understanding root causes to accurately predict failure. Common failure modes are equipment-specific and often depend on process conditions — friction, corrosion, erosion, fatigue, thermal stress, vibration, etc. In addition, equipment performance is influenced by the choice of material, operating environment, manufacturing processes (including variation in process and operating conditions), and underlying structural, fluid mechanics, electrical and chemical considerations.

Detailed engineering simulation that considers all applicable physics complements current reliability and product design practices. It enables parametric, systematic analysis of components and key subsystems in real-life situations. Components and subsystems can be designed, analyzed and validated — across a broad range of physics and operating conditions — to account for uncertainties via computational techniques for virtual prototyping and experimentation. One key tool for success is an integrated framework that enables variation in design across multiple dimensions (geometric scale, physics and domain) and facilitates cross-functional engineering collaboration.

The ANSYS framework and methodologies can help organizations to evolve the use of engineering simulation from a single, siloed design validation practice to a procedure that drives optimization in a world of design input uncertainty. ANSYS solutions combine robust design methodology centered on parametric analysis, design exploration, goal-driven optimization and probabilistic optimization with comprehensive solutions, all applicable to design of advanced material systems, fluid–mechanical systems, electric machine and drive systems, and fluid–thermal systems. ANSYS solutions are ideal for a range of oil and gas industry initiatives. By leveraging the software, teams can streamline product design for reliability, develop reliable subsea systems, and integrate solutions for marine, subsea and offshore structures as well as midstream, downstream, LNG, and FLNG equipment design and optimization. These solutions offer a high degree of certainty that the design is right the first time. This level of high-fidelity analysis and design also benefits flow assurance, wellbore and near-wellbore equipment reliability and projects.

Robust, accurate and proven engineering simulation solutions can be beneficial for evaluation of new concepts and the design of new equipment and facilities. For equipment already in operation, simulation is a powerful tool for identifying failure mode and root causes, driving maintenance schedules, increasing throughput, troubleshooting and establishing fitness for service.

This special issue of *ANSYS Advantage* includes an array of customer case studies and industrial application articles that might inspire you to take even more advantage of engineering simulation. In "In-Depth Solutions," for example, Technip describes how it automates ANSYS software to evaluate over 20,000 simulation runs that ensure subsea jumper pipe structures can survive worst-case scenarios. In "Pipe Dream Becomes Reality," Schlumberger engineers use nonlinear FEA simulation of BHA in a drill string to eliminate lateral buckling. Similar optimization and simulation solutions are used for marine structures in the study titled "A Perfect Fit."

Other articles demonstrate the benefits of ANSYS comprehensive capabilities. The accelerating use of ANSYS engineering simulation solutions throughout the energy industry helps organizations to develop reliable products, increase product performance and reduce environmental impact. They rely on a range of well-established comprehensive solutions covering structural, fluid, electromagnetic and hydrodynamic behavior coupled in an integrated framework. Most important is that by using ANSYS software, oil and gas companies can reduce risk while meeting corporate project and production objectives.

In-Depth Solution

Technip automates evaluation of 20,000 simulation runs to ensure that subsea pipe structures can survive worst-case scenarios.

By Esen Erdemir-Ungor, Design Specialist, Technip, Houston, U.S.A.

Jumpers are piping components of subsea oil production systems that connect one structure to another, such as for linking satellite wells to a manifold, the platform or other equipment. Designing these very important components is difficult because both of the connection points are free to move — within allowable limits — due to thermal expansion, water currents and other factors. Jumper designers need to evaluate every possible combination of movement, expansion and rotation to determine which combination applies the most stress to the jumper, then design the jumper to withstand it.

Technip recently designed four jumpers, each connecting a pipeline end termination (PLET) — the end connecting point of a pipeline — to the manifold of a producing well or another PLET. Technip is a world leader in project management, engineering and construction for the energy industry. With facilities in 48 countries, the company operates a fleet of specialized vessels for pipeline installation and subsea construction.

LOADS ON THE JUMPER

Undersea pipelines are governed by strict codes developed to ensure pipeline integrity to prevent an oil spill. The jumper needs to withstand loads applied to both ends of the pipe while keeping stress in the jumper within the limits specified by the code.

When oil or gas is transported in the pipeline, the pipeline undergoes thermal expansion, and this expansion is transmitted to the jumper. In this Technip application, thermal expansion was calculated to be a maximum of 40 inches in the x-axis and 30 inches in the z-axis. Further displacements of up to 2 inches in the x-, y- and z-axes were possible due to variation when the position of the structures was measured and when the jumper was cut and assembled to its final size. Rotations of up to 5 degrees in either direction in the x- and z-axes were also possible. The net result was a total of three displacements and two rotations on each end of the jumper that needed to be considered at each extreme of its range of motion. To fully understand every load case that could be applied to the jumper, it's necessary to consider every possible combination of these 10 different variables, a total of 1,024 load cases.

Technip engineers had to take into account variability in the position of the PLET and manifold. There is a target location for the two structures, but the position can vary within the project-specified target box. As a result, the length of the jumper can be anywhere from 900 inches to 1,500 inches; furthermore, the gross angle of the jumper with respect to the PLET and manifold also can vary. This

PHOTO COURTESY TECHNIP.

Using conventional analysis tools, it would be impossible for an engineer to solve this many load cases within a normal design cycle.

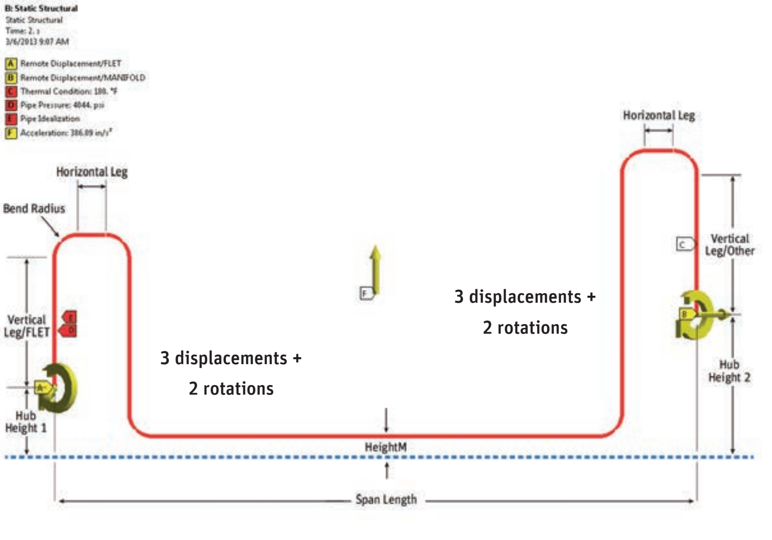
gross angle is important because it determines the angle at which thermal expansion is applied to the jumper. The position of the PLET and manifold are measured prior to jumper installation. The jumper is then cut and welded to the length and angle determined by the measurements just before it is installed. The engineering team addressed these variations by considering four different scenarios for the jumper: maximum length, minimum length, maximum gross angle and minimum gross angle. So a complete evaluation requires that the 1,024 load cases be evaluated for each of these four scenarios, resulting in a total of 4,096 load cases for each jumper design.

Using conventional analysis tools, it would be impossible for an engineer to solve this many load cases within a normal design cycle. The standard practice has been for experienced engineers to use their judgment and instinct to pick out a relatively small number of load cases that they believe will generate the highest level of stress. But operators of wells and pipelines are becoming much more sensitive to potential hazards. In this project, the customer asked that every single load case be evaluated to make certain that the jumper could withstand the absolute worst case. Just a few years ago, such a task would take so long that organizations would rule it out for production jobs. But recent advances in optimization tools now make it possible to rapidly evaluate large numbers of design cases to ensure robustness.

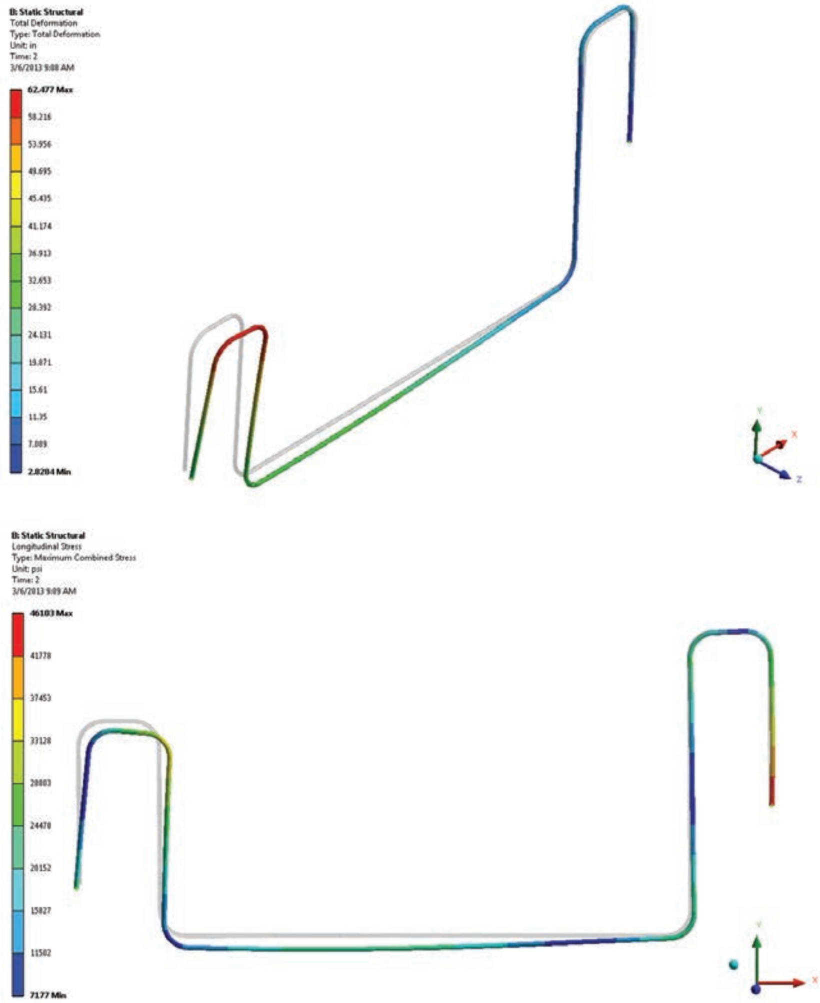
EXPLORING THE DESIGN SPACE

In this project, the first step was to create a simple jumper model in ANSYS DesignModeler based on a previous design. Engineers created three design parameters to define the geometry of the jumper that could be varied to improve its performance. Parameters included the length of two vertical and one horizontal sections of pipe that constitute the core of the jumper (geometric parameters) as well as three displacement and two rotation parameters at each end of the jumper (mechanical parameters), with two possible values representing each extreme end of its range of motion.

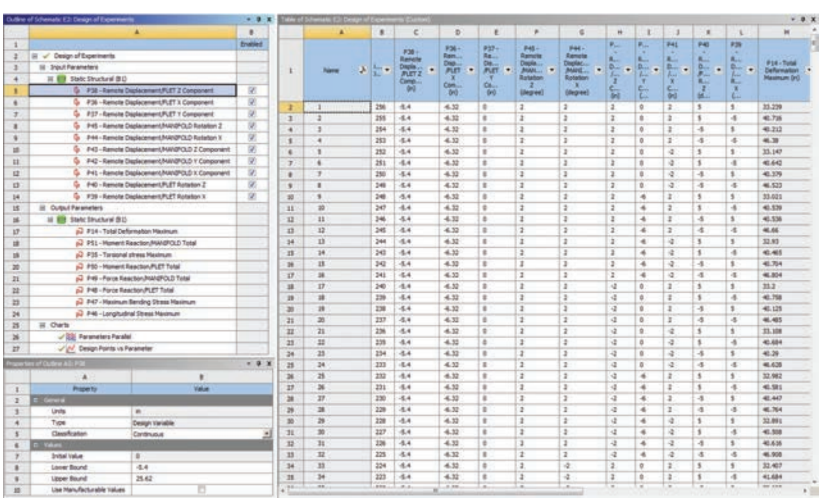
As the first step of the design process, engineers set up a short simulation run to explore the design space. They selected a previous design as the starting point, and the geometric design parameters were allowed to vary over a limited range in increments of 1 foot. Engineers used the Design Points option in ANSYS DesignXplorer to select a subset of about 200 load cases. They



▲ Parameters were allowed to vary during optimization. The diagram shows loads that potentially can be applied to the jumper. Ten variables were applied to the remote displacements.



▲ Total deformation (top) and maximum combined stress (bottom) at true scale



▲ The design of experiments capability in ANSYS DesignXplorer helped the simulation software to run thousands of load-case steps. Actual mechanical parameter ranges are shown.

created a table with these parameters within the DesignXplorer optimization tool. A Technip engineer gave the Update command to solve the model for every combination of values in the table. The first design point, with the first set of parameter values, was sent to the parameter manager in the ANSYS Workbench integration platform. This drove the changes to the model from CAD system to post-processing.

DesignXplorer used parametric persistence to reapply the setup to each combination of parameters while file transfer, boundary conditions, etc., remained persistent during the update. The new design point was simulated, and output results were passed to the design-point table where they were stored. The process continued until all design points were solved to define the design space. The outputs of each simulation run included the minimum and maximum bending stress, shear stress, axial stress and combined stress within the jumper. Technip engineers examined the results, looking particularly at the sensitivity of the outputs with respect to design parameters and whether their variation with respect to the design parameters was linear or nonlinear.

DETERMINING THE WORST-CASE SCENARIO

As the second step, engineers fixed the mechanical parameters at the values that provided the worst results in the previous step with the goal of obtaining the geometric parameter set that could withstand the worst load combinations. Once the mechanical parameters were set at the current worst case (obtained from the first step), then the geometric parameters were allowed to vary over a greater range. Technip created a design-point table using the default settings in the design of experiments. Engineers employed goal-driven optimization for which the primary goals were that the stresses mentioned previously would not exceed allowable values. At the end of the second step, a set of geometric parameters that do not fail under the current worst-case scenario was obtained.

The third step confirmed that the optimized geometric parameter set would not have stresses higher than the allowable values under any possible load combination. Technip engineers created a design-point table using

▲ Jumper installation

the two possible extreme values (minimum and maximum) for each mechanical parameter while fixing the geometric parameters at the values obtained in the previous step. Since there are 10 mechanical parameters, this resulted in 1,024 (2^{10}) load cases. The Custom Design Point table option was used to import the 1,024 determined load cases. Engineers monitored the design-of-experiments runs, and if, for any load combination, the allowable values were exceeded, the design-point update was stopped, then the mechanical parameters that produced high stress were set to the new worst-case scenario. This started the iterative process between the second and the third steps. When all the runs in the third step were completed successfully, so that the allowable values were not exceeded within the pipe and the reactions at the ends of the jumper did not exceed connector limits, engineers moved onto the fourth step.

The 1,024 load combinations for each of the other three scenarios discussed earlier were run using design of experiments for step four. When all the design criteria were met for all 4,096 possible load combinations, engineers deemed the optimized parametric set successful, and the design for the first jumper was finished.

For the second, third and fourth designs, Technip engineers started with the optimal design that had been determined for the first jumper. They ran this design against the 4,096 load cases for each of the other jumpers. The maximum stresses were not exceeded on the last three jumpers — for each jumper design, engineers ran only the 4,096 cases needed to prove that the design could withstand every possible load case.

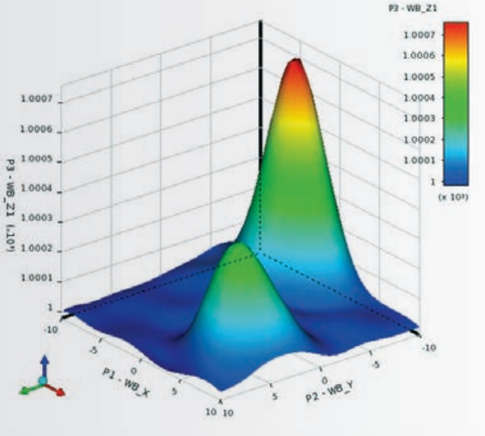
Variations in operating conditions may create uncertainty in subsea pipe structural design. Using parametric exploration and optimization tools from ANSYS, engineers checked the structural performance and integrity of these four jumpers over about 20,000 simulation runs. This capability will provide Technip with the significant competitive advantage of being able to prove to clients that its designs can withstand the worst-possible conditions encountered under the sea.

This capability will provide Technip with the significant competitive advantage of being able to prove to clients that its designs can withstand worst-possible conditions.

Scaling Design Parameters

By Mai Doan, Senior Application Engineer, ANSYS, Inc.

Technip's customer wanted a rigorous study of every possible combination of parameters, and ANSYS DesignXplorer was up to that task. However, many companies employ DesignXplorer to study the design space with as few solved design points as possible. Advanced DOE and optimization algorithms within this tool enable users to choose combinations of parameters that extract the maximum amount of information with minimum resources. Response surfaces (also known as metamodels) interpolate between the solved design points. If, for example, peak loads or optimal designs are predicted between solved design points, these can easily be verified on an as-needed basis. Using automated refinement and adaptive optimization, DesignXplorer focuses solver resources in the areas of the design space that are most likely to yield valuable results.



▲ Response surfaces show the relationships between design parameters and design performance.

PIPE DREAM BECOMES REALITY

Accurate simulation improves reliability and ensures cost-effective deployment of pipe strings in oil wells.

By Jim Filas, Technical Advisor, Schlumberger Rosharon Testing and Subsea Center, Rosharon, Texas

PHOTO COURTESY SCHLUMBERGER.

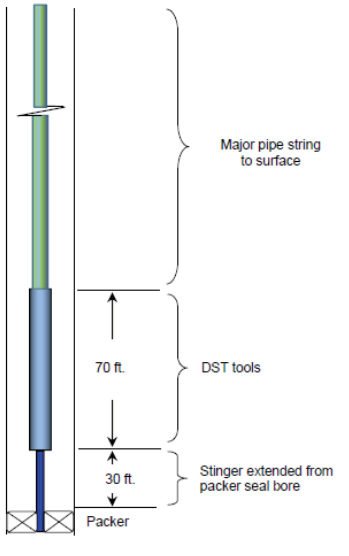
As the search for oil and gas progresses into increasingly deeper waters with exposure to higher downhole pressures and temperatures, accurate prediction of the complex stress state in oil well pipe strings is critical to ensure that operations are carried out safely and efficiently. Accurate nonlinear structural mechanics simulation makes it possible to predict the behavior of the pipe string as it undergoes complex buckling that often results from fluid injection operations. The latest simulation methods enable engineers to design pipe strings and downhole tools with the capabilities to handle the more challenging wells being drilled today.

Schlumberger, the world's leading supplier of technology solutions to the oil and gas industry worldwide, performs well tests to obtain important measurements, such as flow rate and bottomhole pressure, to characterize petroleum reservoirs. During well testing operations, fluids may be pumped under high pressure into the wellbore to stimulate the formation (rock around the borehole).

To conduct a well test, a bottomhole assembly (BHA) consisting of specialized tools and measuring instruments is conveyed downhole on the end of a pipe string and lowered into the well casing. A packer on the BHA's lower end expands within the well to isolate the interior of the pipe string from the annulus between the pipe string and casing. The packer has a smooth internal seal bore that accommodates a seal on the bottom of the BHA. This arrangement permits vertical movement of the lower end of the pipe string while maintaining a seal with the internal diameter of the packer to allow for thermal expansion and contraction of the pipe string during the well test. The lowest stiffness tubular member in the BHA is usually the stinger, whose lower end is fitted with the aforementioned seal assembly and inserted into the packer seal bore. The stinger typically extends out of the top of the packer a distance of 30 feet or more and is joined to the drill-stem test (DST) tools above, which in turn are joined to the major pipe string to the surface.

The lower end of the stinger is free to move vertically, so when the internal string pressure exceeds the pressure in the annulus, a hydraulically induced upward force is applied to the bottom of the stinger. Such conditions exist when fluid is pumped downhole and forced into the formation, such as during hydraulic fracturing or acidizing operations. The resulting pressure wave caused by firing perforating guns [1] can also apply similar upward hydraulic forces.

These upward forces on the bottom of the stinger can cause the BHA and pipe string to helically buckle inside the casing. When helical buckling occurs, as long as the elastic limit of the tubing is not exceeded, the string components will return to their initially straight condition when the pressure difference is removed. However, if the elastic limit is exceeded, permanent corkscrewing of the BHA will result. The stinger, in particular, may become jammed within the casing, preventing the retrieval of the string from the hole and resulting in the loss of expensive tubular assets. In the worst case, this can cause a safety hazard.



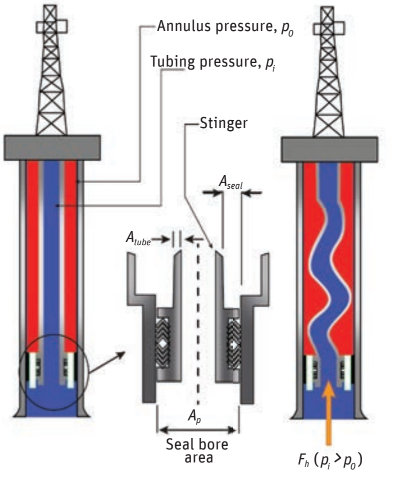
Representative well test string

The latest simulation methods enable engineers to design pipe strings and downhole tools with the capabilities to handle the more challenging wells being drilled today.

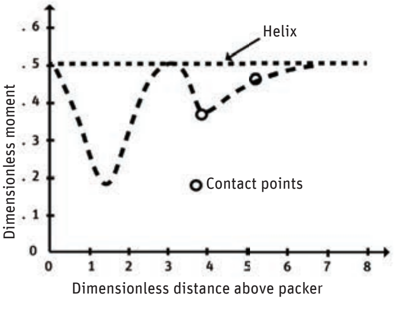
PREVIOUS DESIGN METHODS

Analytical methods have traditionally been used to predict the buckling behavior of the pipe string. First developed by Arthur Lubinski in the 1960s and refined by others in the ensuing decades, these methods not only take into account the hydraulic force acting on the bottom of the string but also the influence of internal and external tubing pressure on the lateral stability of the pipe string. Analytical methods are limited to very simple geometries and typically do not account for geometric nonlinearities, such as large deflections and complex contact between the pipe and casing, so they are unable to accurately predict complex combinations of effects found in the real world. In addition, existing analytical solutions are limited in their ability to accurately predict the post-buckled shape of the packer's stinger in the region where it exits the packer seal bore and before the string fully forms into a helix.

Prior use of finite element analysis (FEA) to study helical buckling required engineers to build large models comprising solid elements. A perfectly symmetrical and straight pipe will not buckle in a numerical simulation, so it is necessary to apply small, random lateral loads to simulate imperfection, and then incrementally increase the bottom load to induce buckling. Lateral deflection is constrained after the pipe contacts the casing wall, and equilibrium is re-established as the pipe forms into a stable helical shape. Modeling a substantial length of the well test string using solid elements and solving a sufficient number of iterative steps to explore the full load range of interest requires enormous computing resources and wall clock time.



Hydraulically induced bottom force



Bending moment in helically buckled pipe predicted by analytical solution

Using BEAM188 Elements for Nonlinear Simulation

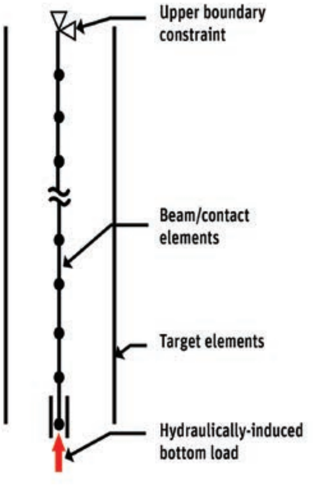
The BEAM188 element has six degrees of freedom at each node and is based on a first-order shear deformation theory that is well suited for linear and nonlinear problems with large rotations and strains. Cross sections remain plane and undistorted after deformation, but they are not required to remain perpendicular to curvature.

Once the lower end of the string buckles and the bottom load is further increased, the model remains continuously unstable as new helix coils are formed progressively higher in the string. For such a highly nonlinear problem, aggressive stabilization control is needed to maintain convergence as the solution progresses. Nonlinear stabilization using pseudo-viscous damping was used to provide the necessary control. Dampers, with appropriate damping coefficients, are attached to each node in the system. At the onset of instability, the integration increment is reduced when divergence of the solution is detected. The damping forces act in the opposite direction of the nodal displacements to enable the solver to obtain a converged solution during what would otherwise be an unstable phase of the simulation.

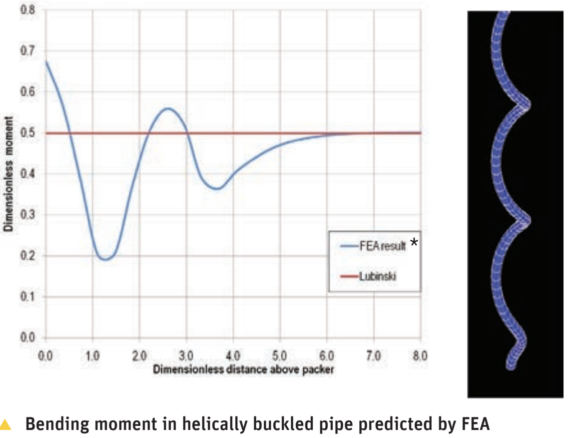
Therefore, the FEA model must be shortened considerably to avoid an impractically large number of degrees of freedom. Limiting the model length in this manner requires imposing boundary conditions, either displacements or forces, where the model is terminated. These boundary conditions are not known before the problem is solved. Applying assumed, imprecise boundary conditions can substantially reduce the accuracy of the simulation.

An alternative approach is to use 3-D beam elements to model a long section of the pipe string in combination with a concentric contact surface to represent the casing constraint. A line element model can be made long enough to represent a significant portion of the pipe string while keeping the model size manageable. However, traditional beam elements based on classical Euler–Bernoulli theory do not work well in this type of model because of inherent restrictions. One of these restrictions is the requirement for the beam cross sections to remain perpendicular to curvature, which prevents the helix from developing after the initial buckling load has been reached.

It is important to more accurately predict buckling and the resulting stress state in the well test string.

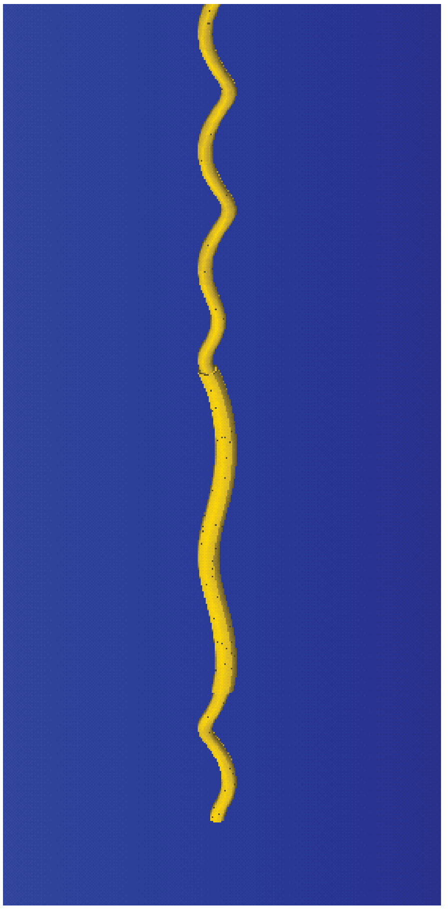


Schematic of finite element method



Bending moment in helically buckled pipe predicted by FEA
*For 3.5-inch continuous string in 9.625-inch casing

The higher accuracy of this FEA method makes it possible to design BHAs that ensure safe operation in deeper wells, which can help satisfy the world's need for oil and gas.



Shape of helically bucked well test string from simulation

The approach provides a more accurate method of simulating helical buckling in a well test string sealed in a packer.

NEW SIMULATION APPROACH

Schlumberger has developed a new approach employing ANSYS Mechanical BEAM188 elements. The team models a sufficiently long portion of the well test string so that the upper boundary condition remains above a sufficiently long portion of the fully developed helix such that the upper boundary condition does not affect the region of interest near the packer. The model is solved as a large deflection, nonlinear buckling problem along a load path that progresses from initial instability and casing contact, to formation of a fully formed helix, and finally to a bottom load magnitude that corresponds to field conditions of practical interest.

To compare the FEA approach with the analytical solution, Schlumberger engineers ran FEA cases for uniform strings of varying diameters constrained by different-sized casings. Unlike analytical solutions, the FEA simulation fully accounts for geometric nonlinearity. Next, the models were enhanced to represent a realistic BHA having regions with different diameters and flexural stiffness. Such an analysis is not practical using a simplified analytical solution.

The new FEA approach provides a more accurate method of simulating helical buckling in a well test string sealed in a packer. For the case of a uniform stinger, there is excellent agreement between established analytical solutions and FEA within the fully developed helix. In the region between the packer and fully developed helix, FEA predicts higher bending stresses than existing analytical solutions. Furthermore, the bending stress in the stinger just above the packer was found to vary somewhat depending upon problem geometry, indicating the influence of geometric nonlinearity on the solution. The discrepancy in the region before the helix is fully developed is attributed to simplifying assumptions inherent in the derivation of the analytical solutions. A nonlinear finite element solution is not restricted to these simplifications. The higher accuracy of the new FEA method will make it possible to design BHAs that ensure safe operation in deeper wells, which can help satisfy the world's need for oil and gas.

Footnote

[1] Perforating gun: downhole device containing explosive charges used to perforate sections of the casing below the packer to allow oil and gas to enter the lower portion of the wellbore

A PERFECT FIT

Researchers develop an automated process for optimizing marine structural components.

By Jouni Lehtinen, Research & Development Engineer, MacGregor Dry Cargo, Kaarina, Finland, and
Sami Pajunen, Associate Professor, and **Ossi Heinonen,** Researcher, Tampere University of Technology, Tampere, Finland

Nearly all marine structural components are custom designed for a specific application. It is also a fact that the highly competitive shipping market has no room for slack. Therefore, advanced ship builders and cargo system suppliers must optimize their structural designs to meet specific application needs. The bottom line: The structure's materials must be in the exact right place — where they best support the needs of the cargo system and enable efficient marine cargo transports. An optimized steel structure with no excess weight translates into optimized and flexible space for transported cargoes.

To address such issues accurately and efficiently, MacGregor Dry Cargo's engineering department and researchers at Tampere University of Technology developed an automated solution for optimizing marine structures; at the same time, the solution ensures that the structures are able to handle the required operating loads. This is done by means of a script that drives an ANSYS template file to perform a finite element analysis (FEA) on a series of design points. The results are used to construct a response surface model (RSM) of the design space. The RSM is reviewed to identify the most efficient design. This process also improves the reliability of the design by reducing the potential risk of design errors.

With the automated optimization solution, MacGregor has a tool that optimizes the process more accurately and efficiently than before.

DEVELOPING NEW, EFFICIENT DESIGN METHODS

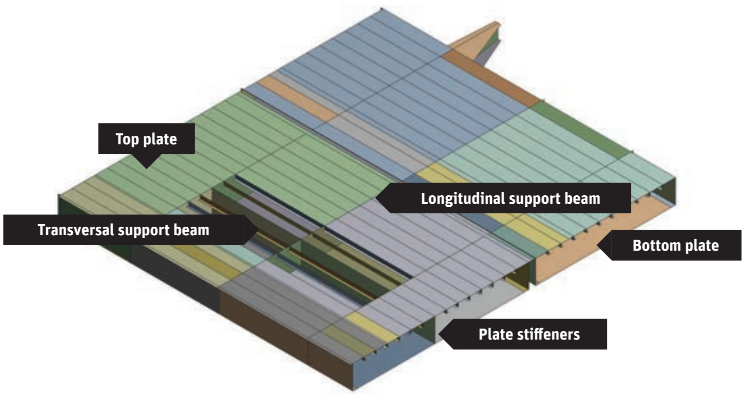
MacGregor offers integrated cargo flow solutions for maritime transportation and offshore industries. The competence center for MacGregor's Dry Cargo business line has a long history of cooperation with Tampere University of Technology, Finland's second-largest university in engineering sciences, for research and development of new design processes and tools.

In this particular marine component application, the team optimized the design to meet specific customer requirements. The cargo profile dictates the basic parameters of the ship's hull design. Within these constraints, the hull should be as light as possible in order to minimize material costs and also to keep the weight of the hull as low as possible. Any weight that is saved in the hull and cargo system design can be used for the benefit of the payload.

Using a standard design in this application — one that isn't optimized to the application — would have increased the amount of material used for the product with no additional value for the customer. Reusing previous designs with similar specifications also can be difficult, because many existing products were customized for project-specific requirements. Furthermore, the traditional approaches do not take advantage of technological advances in parametric design. Another solution for customer-specific optimization — such as simple design rules based on mathematical functions — does not take into account the detailed geometry of the structure, so designs created using this method are less than optimal. The most common method, conventional FEA, has the ability to accurately predict the performance of any single design. However, manual design optimization with FEA requires that a skilled analyst individually study many different models. The high cost and long leadtimes of this process drive up engineering costs. Manual design optimization takes too long to use during the tendering stage, when customers come to MacGregor for a price quote and an initial design to be produced in short turnaround. Finally, assigning experienced analysts to repetitive work is often not the best use of resources.

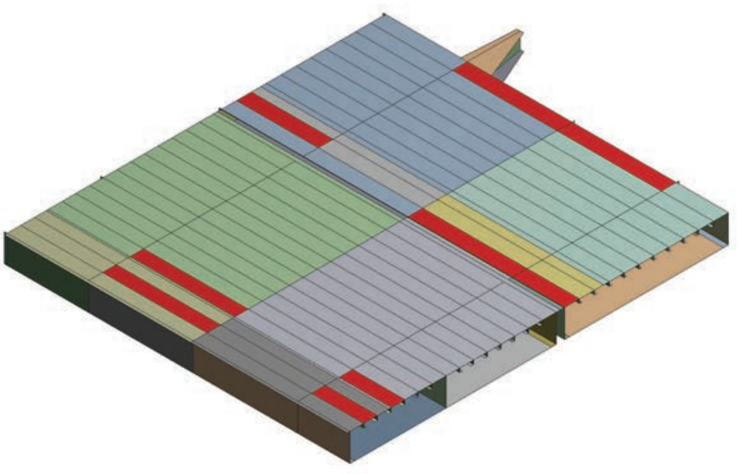
An automated process to optimize marine structures for any application has to address the dimensions and loading of the structure, factors that may vary drastically from project to project. The main components of cargo handling equipment, in this case, are the top plate, longitudinal and transverse support beams, top plate stiffeners and bottom plate. Designers begin the process by creating a parametric mid-surface geometry model in SolidWorks® CAD software. The team uses symmetry

▲ A ship outfitted with a cargo handling system from MacGregor

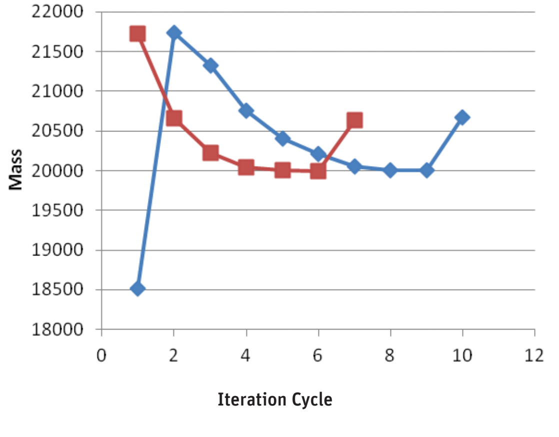


▲ Half of the structure with part of top plate removed to reveal structural members

This process improves design robustness by reducing the potential for design error.



▲ The most critical areas of the top plate, marked in red, are checked for buckling during the optimization process.



▲ Convergence of optimization sequences based on two different initial designs

to reduce the model to half of the structure. To employ this model, the customer provides the main dimensions of the structure during the tendering procedure, and the team enters these values as parameters into the surface model. Designers then parameterize the material thicknesses of the model as the key design variables to be optimized during the automated process.

AUTOMATING FEA MODEL CREATION

An ANSYS Workbench template file that contains ANSYS Parametric Design Language (APDL) commands automatically meshes the model using predefined meshing control settings. The team loads the structure with multiple uniform pressures as determined by the structural codes, and the structure is supported at designated points on the edges. The loads mainly cause compressive stress in the top plate, tensile stress in the bottom plate, and shear stresses on the support beams. The template file generates the load, support and material property definitions. The supports are defined with an APDL command that determines the displacement and rotation of specified nodes. Loads are defined using another APDL command to apply a surface force. The team uses the Named Selections feature to select the nodes and elements for applying the supports and loads. For example, the edges in the symmetry plane are defined as Named Selections and used for locating the supports. Named Selections are also used to define surfaces with the same material thicknesses in selection groups, so the thickness of each element in selection groups can be defined with the APDL commands SECTYPE and SECDATA.

Researchers selected MATLAB® scripts to control the optimization. They used data from previous projects to select an initial design point that reduces the number of iterations required to reach the optimal solution. The scripts generate a D-optimal or Latin hypercube sampling (LHS) design of experiments (DOE) model centered on the initial design point and then drive ANSYS Workbench to create all of the models required to evaluate each run in the DOE model. The results are used to produce an RSM that approximates the complete design space based on the results of a relatively small number of FEA iterations.

OPTIMIZING THE STRUCTURE

The scripts then scan the responses of each output variable in the RSM in a region of interest around the initial point using the simplex algorithm and determine the optimal design. Critical areas in each structural member group are defined as Named Selections so their stresses can be accessed with APDL post-processing commands. The team can easily access the mass of the structure through APDL commands. The optimization minimizes the mass of the structure with respect to constraints based on regulatory codes. The top plate is mostly under compressive and shear stress and, therefore, is constrained against buckling. The beam webs are almost entirely under shear stress and constrained against bucking. The stresses in the bottom plate are constrained so that the material will not yield. The design variables in the optimal design are used to generate another FEA iteration to confirm the RSM prediction.

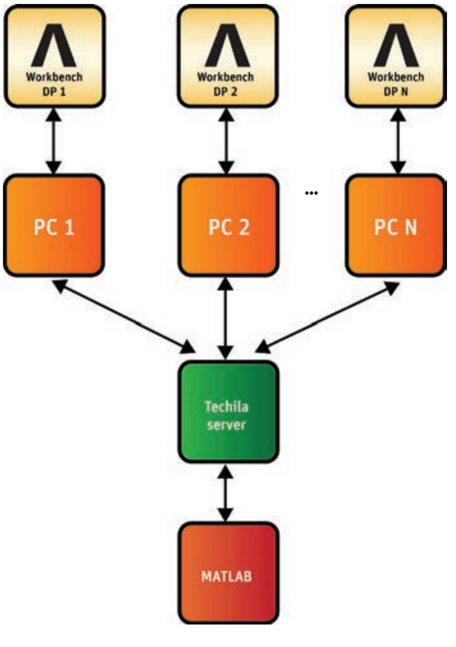
The group tested the automated optimization method by applying it to a stiffened plate structure. The structure is simply supported at support beam ends, and the boxes are connected with bar elements modeling the hinges that constrain the vertical displacements at specific points. The team loaded the structure uniformly on the top panels with a pressure of 45 kPa. The goal was to optimize 17 thickness design variables: eight top plates, four bottom plates, three transverse beams and two longitudinal beam webs. Constraints were derived from empirical knowledge captured from previous projects.

The team optimized the structure using two different initial configurations. The first initial configuration set all design variables at their minimum value as defined by the design rules: 7 mm for the bottom plate and 8 mm for the other parts. The other initial configuration set all variables equal to 9 mm. The optimization process ended when the relative change of mass from one iteration to the next was less than 1 percent. Both of the initial design points resulted in similar objective and constraint function convergence. The optimization process took about six hours using a desktop computer with a single core.

The researchers migrated the optimization process to a Techila Technologies Ltd. high-performance computing system cluster. The 18 design points used to create the RSM were run in parallel instead of sequentially, thus significantly reducing the time of each iterative optimization round.

This project demonstrates that design of marine structures can be quickly and inexpensively automated by constructing an RSM based on successive FEA analysis runs. The automated process optimizes the design in much less time than would be required by an analyst performing the same task manually. With this method, MacGregor can respond to a customer inquiry with a speed and accuracy that did not exist before, and with a design that is optimized for each specific application.

Development work has been supported by Finnish Metals and Engineering Competence Cluster (FIMECC).



▲ Optimization process

MacGregor can respond to a customer inquiry with a speed and accuracy that has not existed before, and with a design that has been optimized for each specific application.

DESIGNING FOR REAL-WORLD REPAIRS

Linear and nonlinear structural analyses improve pipeline repair using composites materials.

By Eri Vokshi, Civil Engineer, Neptune Research, Inc., Lake Park, U.S.A.

When a commercial pipeline needs to be repaired, there is no room for second guesses. Industrial oil and gas pipes can break down due to damage, corrosion or both. When they do fail, many thousands of utility customers may be left without service. Furthermore, there could be significant risks, to both the surrounding environment and public safety. During such an emergency, composites materials have proven to be critical in making necessary repairs. Repairing personnel require a reliable system that can handle extreme real-world conditions and can be deployed in a timely fashion. For manufacturers of composites pipe repair systems, such as Neptune Research, Inc. (NRI), structural analysis from ANSYS plays a critical role in this effort.

At NRI's Florida headquarters, common practice is to develop virtual prototypes before building physical ones. NRI engineers subject a computational model of a physical project to various load cases to see how the repair system responds. The ANSYS suite of structural analysis solutions is ideal for such applications, taking into account loads like deformations, vibration characteristics and reaction forces. To satisfy customers' expectations, NRI's repair systems need to perform at harsh ground conditions, including extremes in temperature and precipitation, and underwater. In meeting these

NRI's repair systems need to perform at harsh ground conditions, including extremes in temperature and precipitation, and underwater.

▲ **Exposed pipe elbow repaired with FRP composites material**

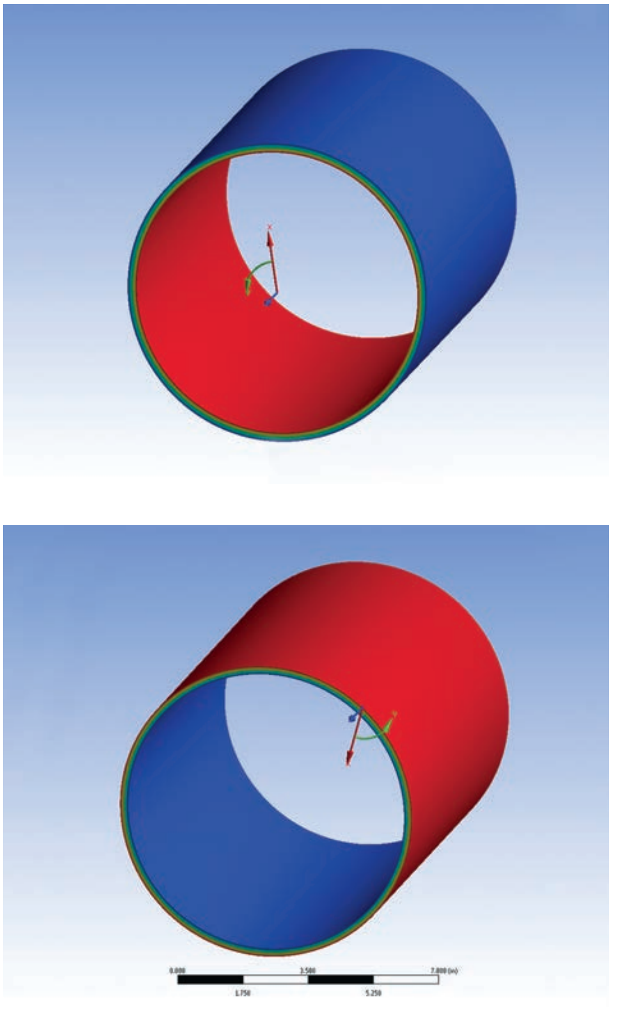
performance requirements, NRI designers use ANSYS Composite PrepPost and ANSYS Structural to compare linear and nonlinear analyses of steel pipe and to help determine the effectiveness of repairs using fiber-reinforced polymers.

In many applications of pipe reinforcement, deterioration of pipe-wall material due to corrosion or other physical damage is assumed to be large, which means greater than 50 percent. At such wall deterioration, the designed internal pressure for an undamaged pipe often produces stress that exceeds the yield strength of the remaining steel. Thus, external application of a fiber-reinforced polymer (FRP) composites material is needed to reinforce the weakened section of pipe. Simulation with ANSYS tools has helped the NRI team to predict and verify the performance of a repair solution. In this case, the repair solution that was analyzed was NRI's Viper-Skin™ system.

For the nonlinear simulation of a reinforced pipe, the NRI team used ANSYS Composite PrepPost. The steel was modeled as an elastic, perfectly plastic material. The yield and ultimate strengths of the pipe were 43,000 psi and 65,000 psi, respectively. Material properties were obtained from the steel-mill certificate (which certifies the manufacturing standards of the mill's product). For the linear simulation, NRI used an isotropic material with a defined modulus of elasticity and Poisson's ratio. For both linear and nonlinear pipe simulations, the team modeled the Viper-Skin composite as an orthotropic material with a linear stress–strain curve. Material properties of Viper-Skin were determined from third-party testing and internal materials testing.

NRI began its simulation study by using ANSYS Structural to analyze an undamaged pipe with an internal pressure of 5,700 psi, which was the burst pressure observed from hydrostatic testing. Results from the linear simulation indicated that the inside surface of the pipe was more highly stressed than the outside surface, while the nonlinear simulation showed that the outside surface was more highly stressed. The stresses produced by the burst pressure are beyond the steel's yield stress, so comparing those stresses to the results from a linear stress analysis is not valid. A nonlinear stress analysis is needed for the comparison.

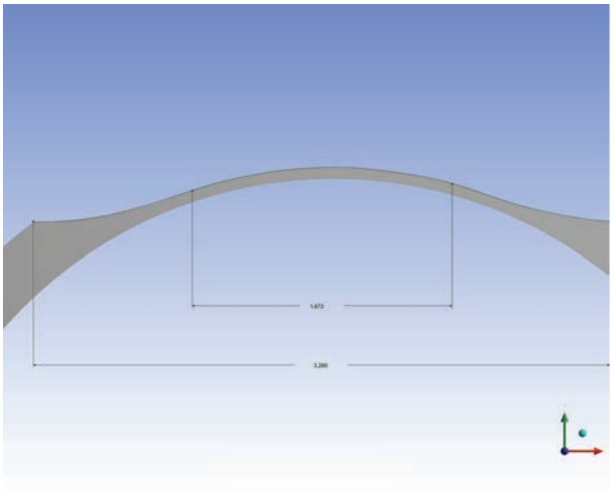
In the next step of the simulation, NRI's engineers introduced an external defect into the pipe representing 80 percent wall loss while maintaining an internal pressure of 5,700 psi. As part of the repair system, the plan involved filling the physical defect with a proprietary epoxy to optimize load transfer between the defect and the Viper-Skin. In Composite PrepPost, the team used an isotropic material to represent the epoxy filler. Material properties of the epoxy were determined from third-party testing and internal materials testing. Without repair, ductile yielding would eventually lead to premature rupture of the steel. Simulation predicted that the repair would hold solid even with the 80 percent wall loss, and subsequent physical testing confirmed the validity of the ANSYS model.



▲ **Pipe hoop stresses in undamaged steel pipe using linear material model (top). The maximum stress is approximately 65,000 psi. Pipe hoop stress modeled using nonlinear material model for steel (bottom), subject to the same load. The maximum hoop stress is approximately 63,000 psi.**

The NRI team found that nonlinear analysis capabilities of ANSYS structural mechanics tools combined with Composite PrepPost were very useful in predicting stress distributions in composites-reinforced pipes. The flexibility of ANSYS software allowed the NRI engineering team to capture the subtleties of dealing with the properties of composites materials. Although the research team could perform its job without simulation, ANSYS software gave the typical user the ability both to respond more quickly and to take more details into account. For example, engineers could analyze the predicted behavior

Nonlinear analysis capabilities of ANSYS structural mechanics tools combined with Composite PrepPost were useful in predicting stress distributions in composites-reinforced pipes.



▲ Typical defect profile simulated by NRI engineers (left). Eighty percent of the pipe wall is removed over an area 3.28 inches long by 1.67 inches wide. The simulation results were then compared against physical testing of the same defect machined into a pipe (right).

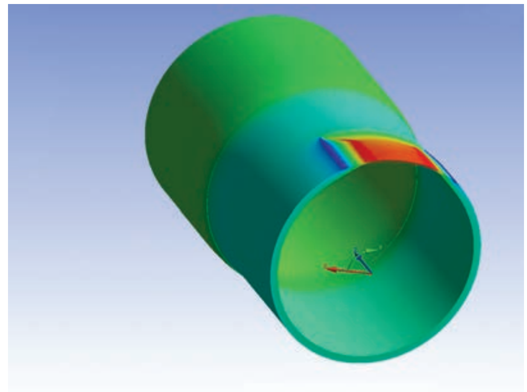
of different composites materials side by side within minutes. Thought of in another way, you could walk to the store and carry groceries home on foot, but driving is faster and allows you to transport more items. Similarly, simulation has enabled NRI's engineers to be more productive.

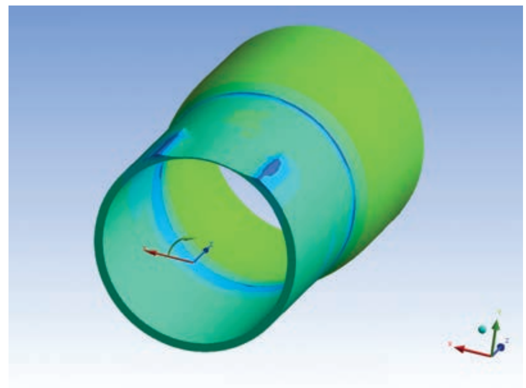
ANSYS tools have proven to be instrumental in making sure that demanding oil and gas industry customers obtain the quality repair solutions that they rely on from NRI. In addition to using the software for detailed linear and nonlinear analyses, NRI engineers apply harmonic analysis (for determining harmonically time-varying load responses) and spectrum analysis (for random vibrations). Beyond that, the team is working on composites flaw detection models to evaluate the effects of various types of damage and their impact on pipe load capacity.

These types of studies aid a company in reducing its product development cycle while improving performance. This can result in a significant advantage over competitors, and ANSYS simulation tools have provided NRI with that advantage. Being quick to market is important, but the repair solution also must be easy to use and reliable. More important is that simulation continues to give NRI designers confidence that their repair designs are sound — before they are put to the test in the real world.

Simulation gives NRI designers confidence that their repair designs are sound, well before they are put to the test in the real world.

▲ Actual physical test pipe specimen pressurized until failure. As predicted with ANSYS Composite PrepPost analysis, the repair held solid. This test confirmed the validity of the simulation model and the strength of the composites repair.





▶ Stress distribution in a repaired pipe. Using linear analysis (top), results show the peak stress to be within the flaw, incorrectly suggesting that the repaired pipe would still fail. Using nonlinear analysis (bottom), there is no stress concentration within the flaw area and the pressure load is uniformly distributed, which is a desired condition for pipe repair.

DESIGNING SOLID COMPOSITES

Employing ANSYS Workbench workflow streamlines simulation of solid composites.

By Matthias Alberts, CEO, CADFEM US Inc., Greenville, U.S.A., and
Pierre Thieffry, Lead Product Manager, ANSYS, Inc.

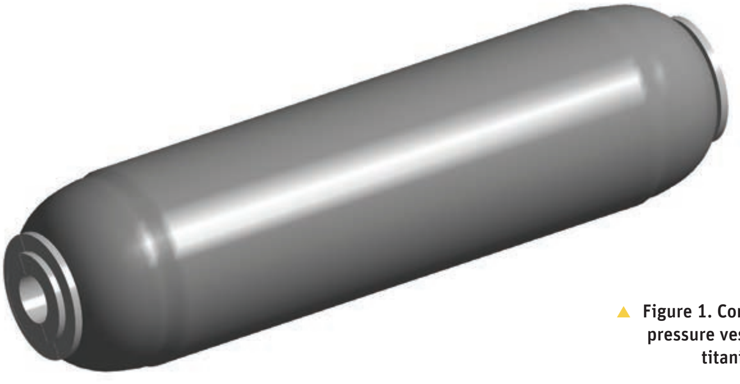


Figure 1. Composites pressure vessel with titanium caps

Traditionally, layered composites structures are modeled as thin structures using shell elements. This approach is valid when designing thin parts, such as hollow tubes for bikes, panels for airframes and wind turbine blades. But when the parts are more massive, such as gas turbine blades or stringers for pressure vessels, using shell elements is not appropriate. In such cases, both stresses in the direction of the thickness and shear stresses out of plane are significant, and solid models are required. Solid models are also appropriate when loads are applied in the direction of the thickness or when the structure is subject to large deformations.

While defining thin-layered composites poses several challenges, the definition of solid composites is even more complex. The shapes usually are not simple and require special treatment — a turbine blade, for example. Composites products generally include noncomposites parts that must be included in the simulation. Consequently, the engineering team needs an efficient workflow for the design of products made of layered solid composites and other parts. An effective process starts by examining the layer definition, based on the same method as used for thin structures, then moves on to create solid composites by extrusion. This is followed by the assembly of composites and noncomposites parts, culminating in analysis of potential failure of the overall structure.

To highlight this workflow, the example presented is a pressure vessel (Figure 1). The entire simulation process is performed in ANSYS Workbench (Figure 2) using ANSYS Composite PrepPost (ACP). The workflow begins by defining the geometry. The model is split into shell composites parts (A) and solid noncomposites parts (D), which are recombined as solids to create the final description of the analysis (B). This combined solid assembly includes connections between parts, loads and boundary conditions, as well as results such as stresses or deformations. The investigation of composites failure occurs as the last step in this process (C).

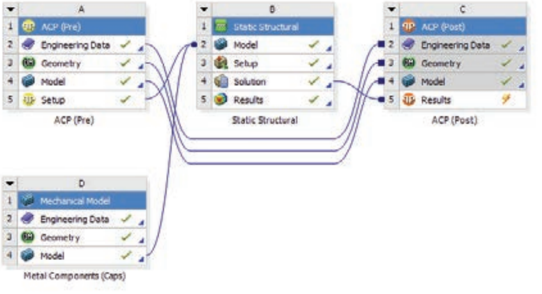
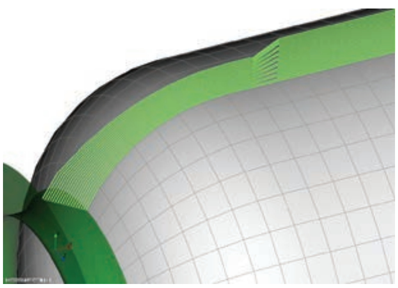


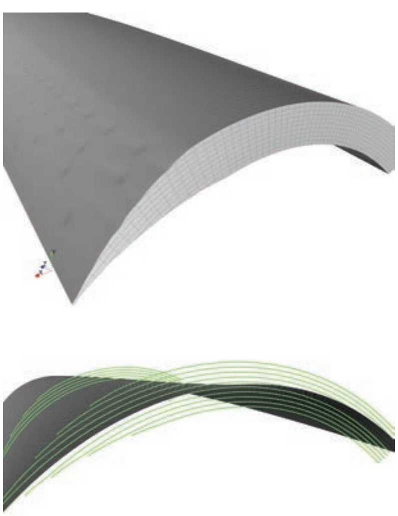
Figure 2. Workflow for design of a composites pressure vessel

The entire simulation process is based on a workflow in ANSYS Workbench using ANSYS Composite PrepPost.

To start, define the layers on a surface mesh (Figure 3). The surface generally will be the inner or outer surface of the product being designed. Define the layers in the following sequence: definition of materials, fabrics (and, optionally, stackups), orientation of the various surfaces of the composites (possibly including draping for highly curved surfaces) and, finally, ply sequence. This approach is very close to the actual manufacturing process. An analogy can be made between the initial surface and a mold, and the ply sequence defined within the simulation tool can be the same as the actual fabric layup within the mold. However, the simulation tool obviously offers more flexibility in ply ordering, as plies can be swapped easily, modified or removed to achieve the required stiffness, weight and cost requirements.



▲ **Figure 3. Definition of ply sequence on inner surface of vessel**



▲ **Figure 3a. Ply tapering**

The critical area in the creation of solid composites is generating a solid model of the layup using a solid extrusion, based on the previous surface definition of the plies. Advanced capabilities, such as ply tapering, surface smoothing or extrusion guidelines, are available to deal with complex shapes (Figures 3a, 3b and 3c). You may apply ply tapering using cutoff rules. Surface smoothing can be performed using the snap-to-geometry feature to fit the extruded model to a given CAD surface. Extrusion guidelines help to extrude the surface model along arbitrary directions.

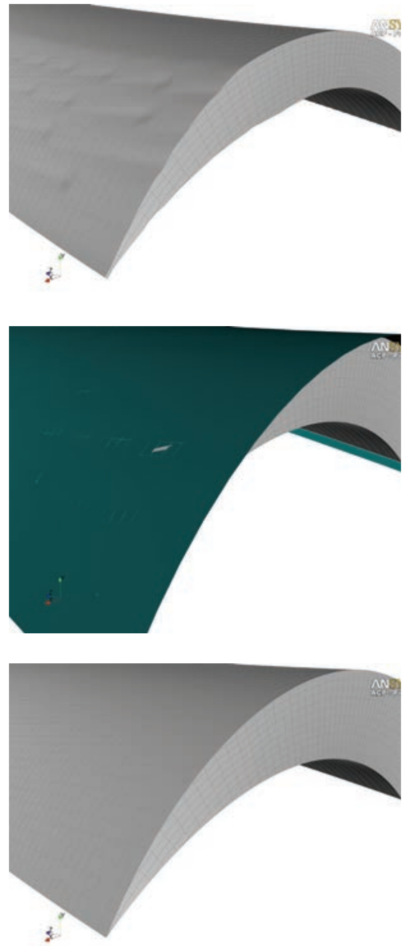
Another important aspect is handling drop-offs. Ply drop-offs can cause damage and delamination in a composite layup. In simulation models, drop-off elements are represented by degenerated brick elements. They usually are made of a homogenous material such as resin.

Once the solid model has been created, it is automatically merged into the final assembly along with the noncomposites parts (Figure 4). Automated contact detection between parts, loads and boundary conditions, and solution settings all can be specified as you normally do for any regular model in ANSYS Mechanical. The transfer of the composites material definition in the full assembly is completely automated. Once the model has been solved, standard results such as deformations or stresses can be displayed on the full model.

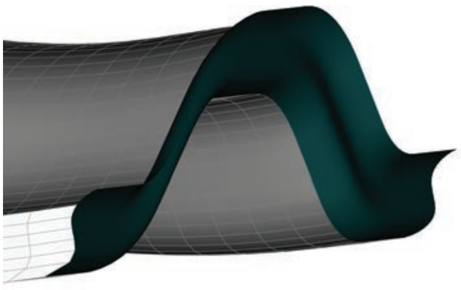
The transfer of the composites material definition in the full assembly is completely automated.

Additional capabilities are available to analyze potential failure of the composites parts. Three-dimensional failure criteria (maximum stress, maximum strain, Tsai–Wu, Tsai–Hill, Puck, Hashin, Cuntze) are available. A typical failure plot gives the analyst information on the risk of failure and the potentially problematic elements and layers in the assembly (Figure 6). You can even create a graph to show failure criteria values through all of the layers at a given location on the model.

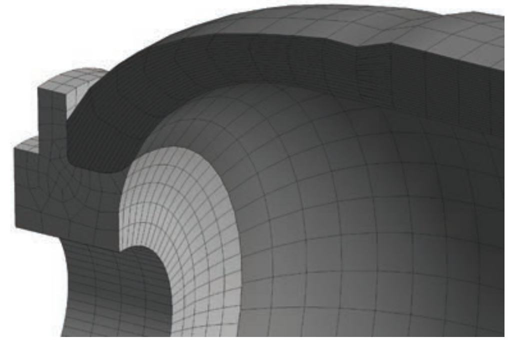
Employing the Workbench-based workflow for solid composites delivers benefits from additional capabilities. If the composites parts are subject to pressure from a flow environment, you can easily add the fluid flow simulation to the simulation, and pressures can be mapped automatically on the structural model. Users also have the ability to parameterize a model to perform sensitivity or optimization studies based on geometry or composites variations (thicknesses, orientation, etc.) using the failure criteria as a performance indicator of the design.



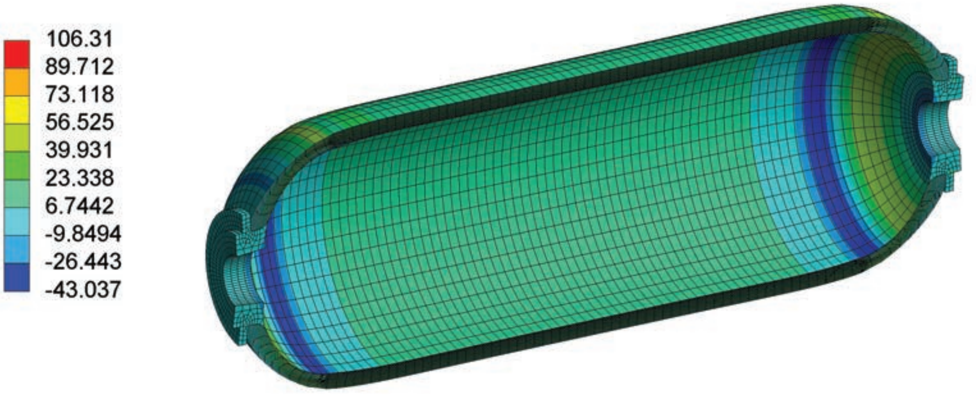
▲ **Figure 3b. Surface smoothing using CAD surface (in green)**



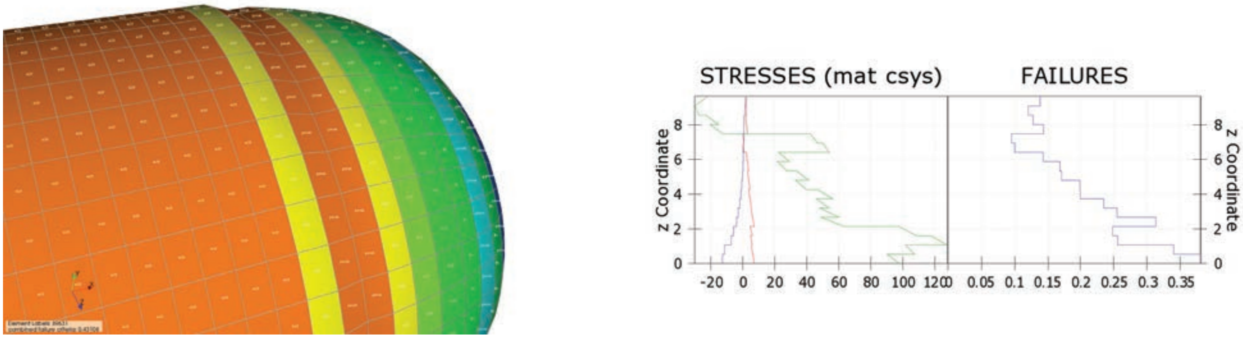
▲ Figure 3c. Extrusion guidelines



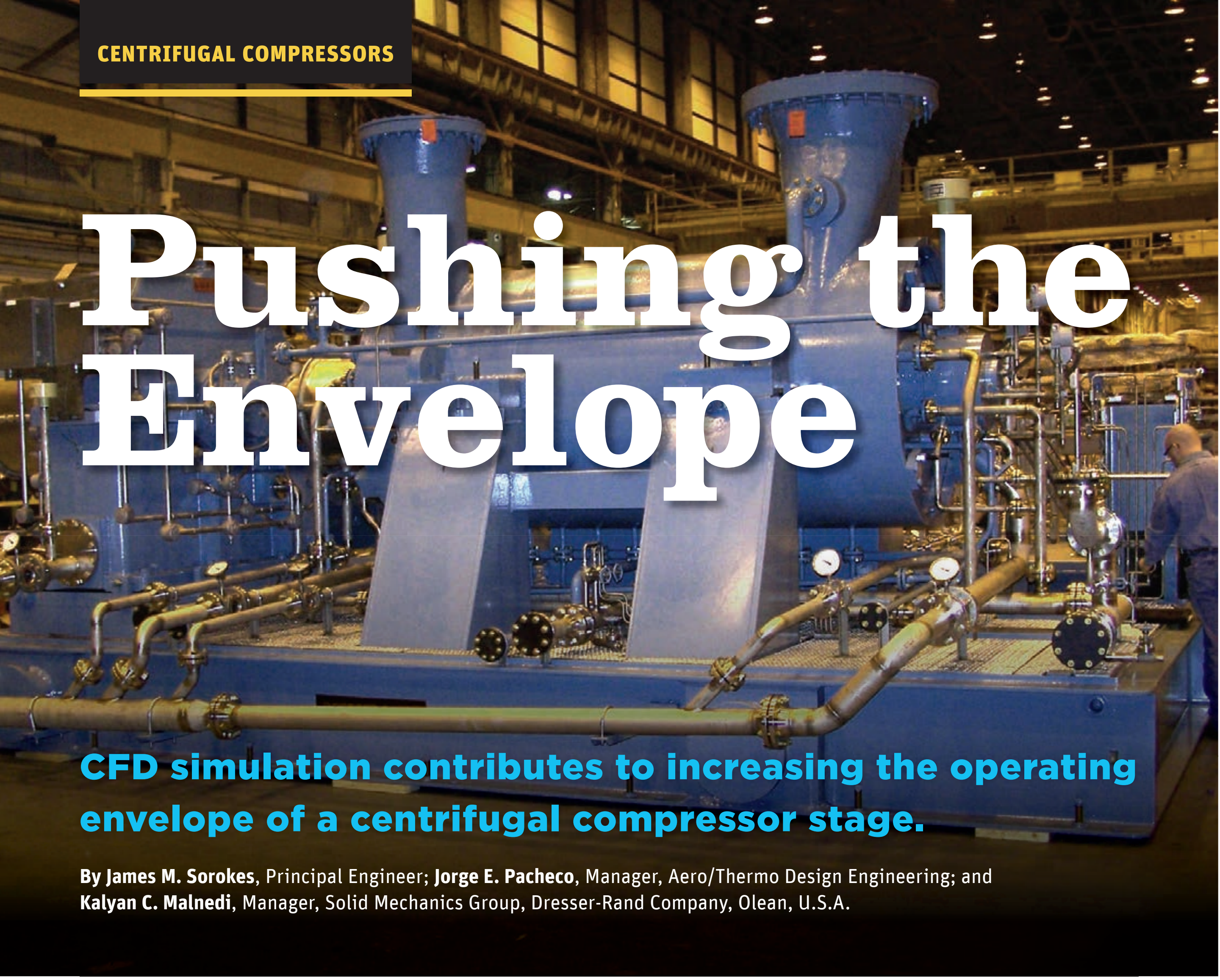
▲ Figure 4. Assembled model showing composites and noncomposites parts (metallic caps in light gray)



▲ Figure 5. Stresses on assembled model



▲ Figure 6. Post-processing composites showing failure on full model (left) or through element layers (right) in charts



Pushing the Envelope

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

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

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

COMPRESSOR DESIGN CHALLENGES

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

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

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

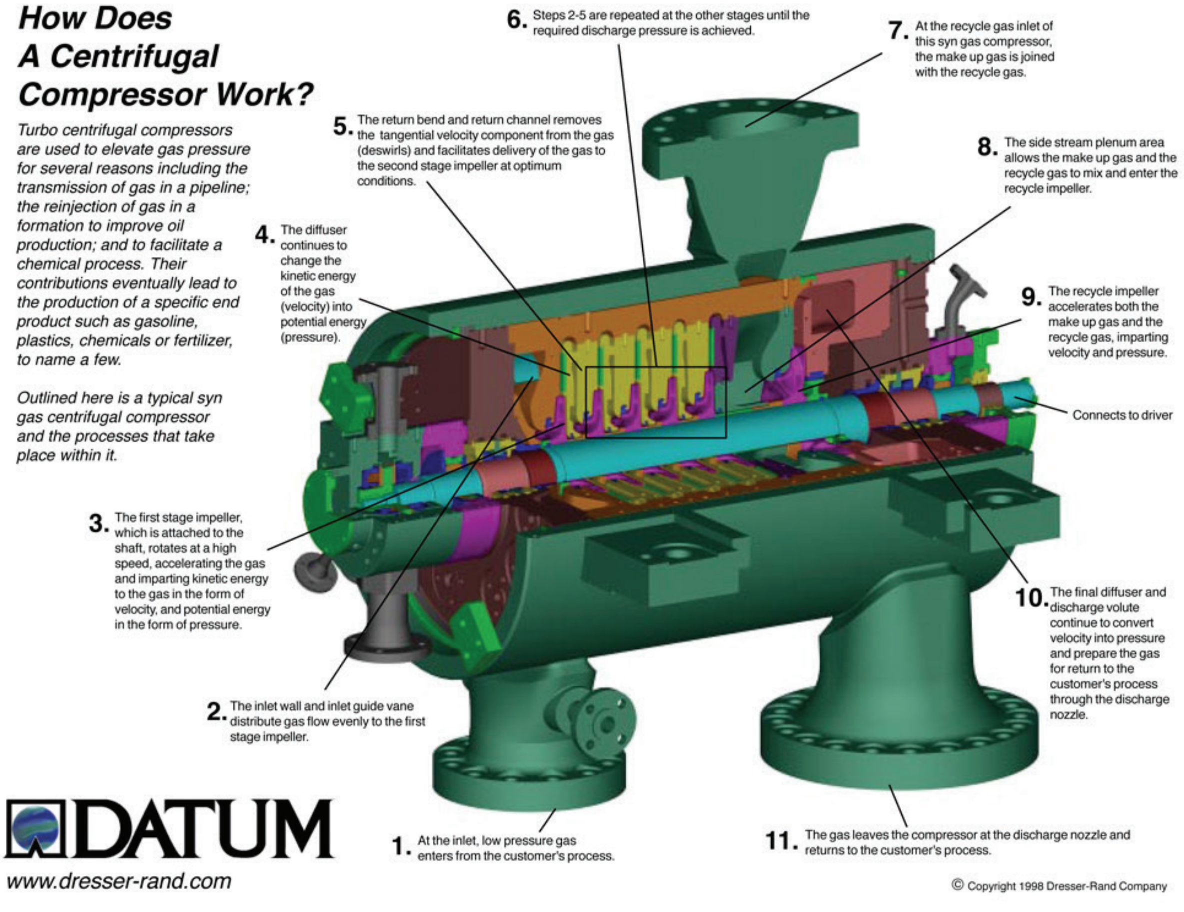
Two factors limit the overall operating range of a compressor: surge or stall margin, and overload capacity. Surge or stall margin limits the compressor's ability to operate at flow rates lower than design, while overload capacity limits the ability to operate at higher rates. Rotating stall arises when small regions of low momentum or low pressure (referred to as stall cells) form in the flow passages and begin to rotate around the circumference of the compressor. These flow and/or pressure disturbances cause unbalanced forces on the compressor rotor, leading to unwanted vibration issues and reduced compressor performance. Surge occurs when the compressor is no longer able to overcome the pressure in the downstream piping and pressure vessels, and the flow is forced backward through the compressor.

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

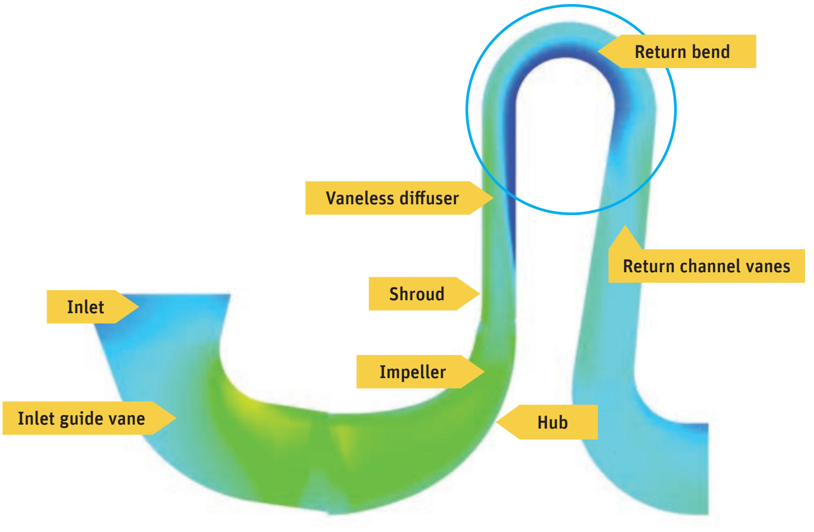
USING CFD TO OPTIMIZE THE DESIGN

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

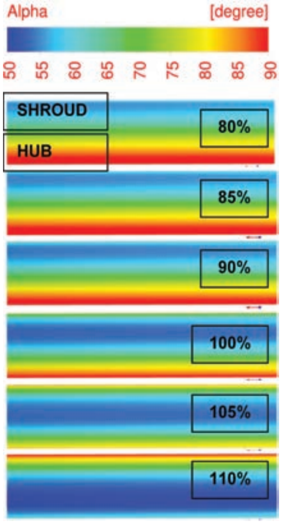
The team evaluated several combinations of pinch, shroud-tapered and



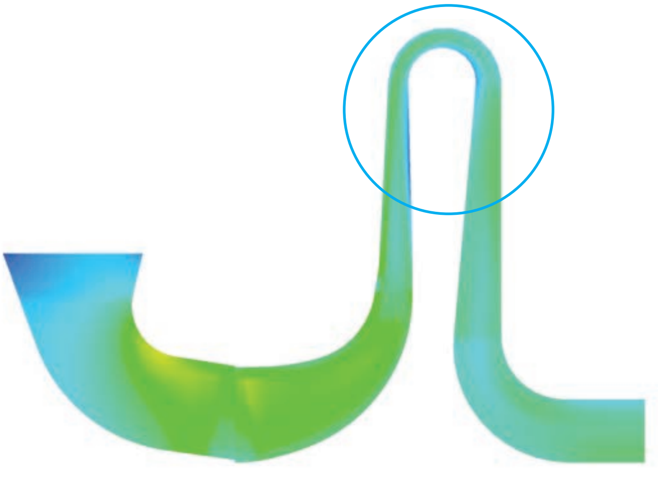
▲ Centrifugal compressors operate by adding velocity pressure or kinetic energy to the fluid stream and then converting that kinetic energy into potential energy in the form of static pressure. Kinetic energy is added by rotating impellers, while the conversion of velocity pressure to static pressure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



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



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



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

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

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

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

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

better pressure recovery at lower flow. The CFD results predicted an improvement in surge margin of approximately 15 percent.

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

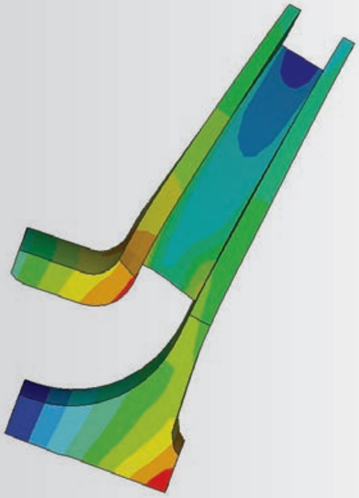
Impellers are subjected to inlet and exit flow variations through the stage, and therefore they must be designed to withstand the alternating pressure loads due to these variations in addition to withstanding steady loads. The structural team used ANSYS Mechanical software and in-house tools to ensure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

Dresser-Rand's use of CFD simulation to optimize the stationary components of a new centrifugal compressor design accomplished several goals: This new design delayed the trans-

Structural Analysis

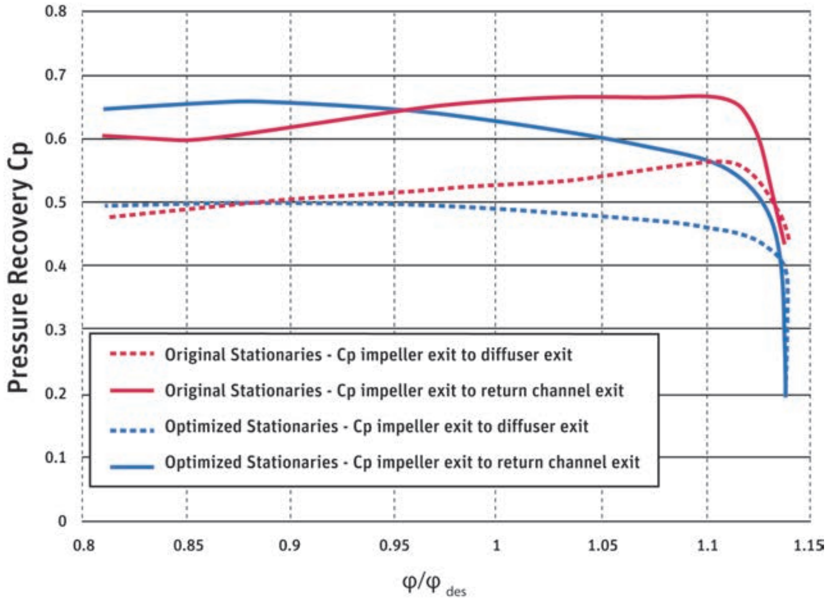
Dresser-Rand structural engineers optimize the impeller design to keep the static stresses both below those seen in similar families of impellers and below allowable material yield strengths. The lower the stresses, the faster the impeller can be run. During the design process, engineers also analyze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

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

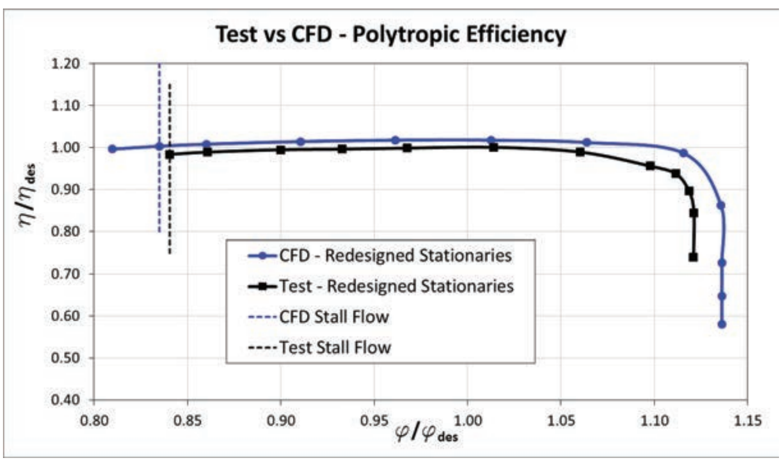


▲ **Typical steady-state plot of impeller**

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



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



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

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

▲ Testing shows that optimized design improves the operating range.

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

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

References

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

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

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

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

Raising the Standards

Fluid-mechanical simulation can help prevent offshore disasters by supporting development of more effective structural standards.

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

The critical nature of offshore structures has been tragically demonstrated by incidents such as the Piper Alpha explosion in the North Sea in 1988, which took 167 lives, and the Deepwater Horizon fire in 2010 in the Gulf of Mexico, which caused both loss of life and environmental damage. Every offshore disaster provides an opportunity to understand its causes and to ensure that future structures are designed to avoid its recurrence. Computer simulation can play an important role by helping to diagnose real-life or potential disasters and evaluate the effectiveness of alternate remediation methods.

Offshore oil and gas production in Canada started in the late 1980s with the development of the Cohasset and Panuke fields (first oil in 1992) off the coast of Nova Scotia, followed by the Hibernia field (first oil in 1997) off Newfoundland. Atlantic Canada's offshore presents one of the harshest weather environments in the world. Oil-related tragedies already have occurred in this area, such as the 1982 sinking of the Ocean Ranger mobile offshore drilling unit, with complete loss of life at the Hibernia field.

AVOIDING REPEAT MISTAKES

Many of the offshore disasters that occurred throughout the world could have been prevented if only the best practices

Richard Grant has worked on developing structural standards for offshore platforms since 1997, when he became a founding member of the Canadian Advisory Committee (CAC) on Offshore Structures Standards under the Standards Council of Canada (SCC). His involvement with the CAC began with providing input into the offshore structural standards then being developed by the International Organization for Standardization (ISO). Shortly afterward, while working on a Canadian offshore project, Grant noted serious shortcomings in Canadian regulations and standards pertaining to fire and explosion safety. He has since been instrumental in correcting deficiencies in Canadian offshore standards through his work with the Canadian Standards Association (CSA). He was subsequently called upon by the international community to assist with the related work being performed under ISO.

Simulation can play an important role by helping to diagnose real-life or potential disasters and evaluate the effectiveness of alternate remediation methods.

available at the time had been followed. During development of the Sable and Terra Nova offshore projects in Atlantic Canada, it was recognized that Canada's CSA offshore standards needed to be reviewed and, where necessary, updated. It was also recognized that Canada's efforts in this area would be better served through direct participation in developing the new offshore structures standards being established under ISO. International standardization of industry best practices helps ensure that lessons learned are captured, and past mistakes are not repeated.

The offshore environment's nature means that realistic physical testing is often too expensive or dangerous and, in many cases, is simply impossible. Simulation plays a crucial role by enabling those involved in developing standards, as well as those designing offshore structures, to evaluate potential disaster scenarios and determine the impact of implementing requirements to help improve the safety of workers and the environment.

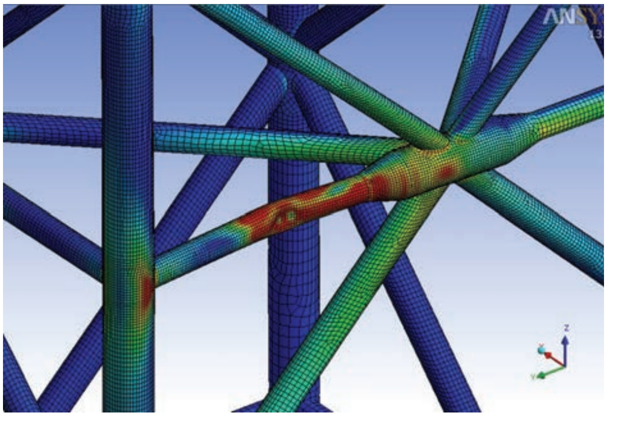
SHIP-TO-PLATFORM COLLISION

ANSYS Mechanical structural simulation has been used to gain a better understanding of one of the more dangerous offshore scenarios: a ship-to-platform collision. The 2005 incident at Bombay (Mumbai) High North, off the coast of India, is a good example for illustrating this situation. A vessel was called to the platform to transport an injured person to shore for medical attention. The vessel came too close to the platform, and it hit and ruptured risers carrying gas to the platform. The subsequent fire destroyed the platform and resulted in many fatalities.

To avoid such catastrophes, offshore structures are required to safely protect critical components such as risers and to absorb energy during a collision. This collision energy is dependent on the vessels permitted within the safety zone around the platform. In performing a simulation of this type, a structure is typically modeled using shell elements with nonlinear material properties and large displacements to accurately represent the resistance of the structure. Collision causes tubular denting, leading to large plastic strains. The structure absorbs energy as its tubular members are crushed by the impact of the vessel. Analysis can accurately capture the amount of energy that the structure can absorb. This type of simulation is invaluable to assess the safety of offshore platforms and can be used to assure that they can withstand certain types of collisions without catastrophic failure, as required by the standards.

PROCESS PRESSURE VESSEL INTEGRITY

Grantec is performing research pertaining to the integrity of process vessels subjected to external hydrocarbon gas explosions. This research is aimed at better understanding the



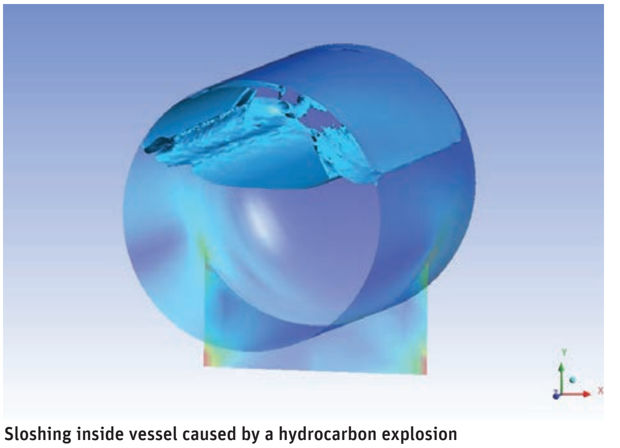
Analysis is used to confirm that the structure can absorb sufficient energy to withstand impact from ships permitted within the safety zone.

Progress in Canada's Offshore Standards

Canada's own offshore standards were significantly improved in the area of fire and explosion safety through new provisions in the CSA Offshore Structure Standards (2004). With Canadian input, similar provisions were developed for the ISO 19901-3 Topsides offshore structures standard. The ISO standards will improve safety in offshore structures around the world. Canada also holds leadership positions in ice loading and concrete construction requirements for offshore structures; the country's experts have provided significant input in these areas of the ISO standards.

Lessons learned from the international community and from Canadian projects have been considered in the rewrite of the Canadian offshore structures welding requirements, contained in the CSA W59-13 welding standard, which is referenced by the ISO 19902 standard for Fixed Steel Structures. Canada's participation with international standards bodies benefits Canada by ensuring that the country's standards reflect the advances made by the international community; it also helps to ensure that advancements made in Canada are reflected in international standards. Many of the new ISO offshore structures standards have been adopted as National Standards of Canada, replacing the CSA S47x Canadian offshore structures standards.

The nature of the offshore environment means that realistic physical testing is often too expensive or dangerous, and in many cases is simply impossible.



Sloshing inside vessel caused by a hydrocarbon explosion

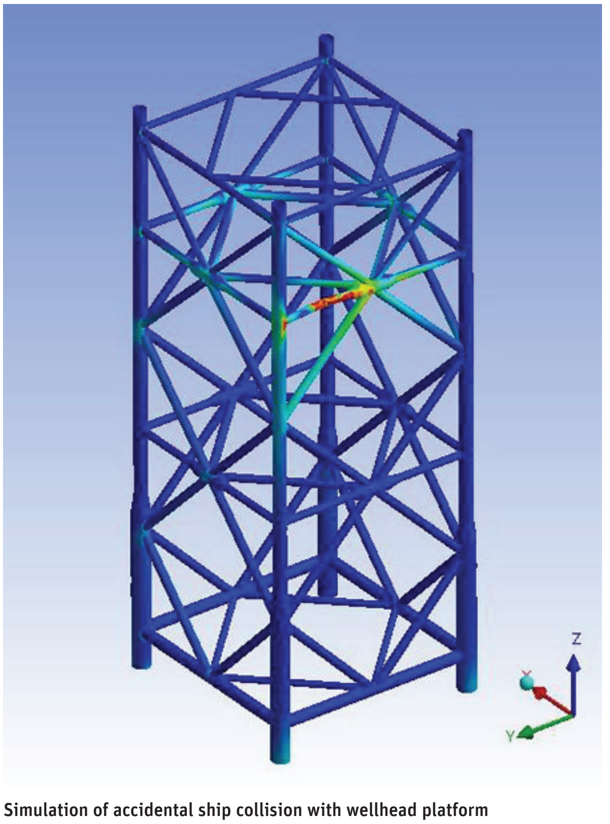
dynamic response of the process vessel shell (pressure envelope), including effects due to internal fluids and the support structure, such as saddles, process skids and decks. Rupture of the pressure envelope of a hydrocarbon process vessel, or the failure of vessel supports, during a hydrocarbon explosion can cause the event to escalate. In the case of a rupture, more hydrocarbons are released to fuel a fire, and, in the case of support failure, the vessel could become a projectile impacting and impairing other safety-critical systems. In the Piper Alpha disaster, failure of hydrocarbon process piping and equipment resulted in the escalation of the fire onboard the platform. In this incident, a loose blind flange on piping resulted in a gas leak that, in turn, produced an explosion that tore loose a bulkhead. The explosion launched the bulkhead, which then ruptured process piping and equipment, resulting in a fire that continued to escalate and destroy the platform with significant loss of life.

Process vessel integrity research at Grantec is being performed using multiphysics simulation software from ANSYS. The simulation is a fully coupled transient fluid–structure interaction (FSI) analysis that uses ANSYS Mechanical for the structural simulation and ANSYS CFX for the computational fluid dynamics (CFD) analysis. The vessel shell and saddle supports are modeled with shell elements, and the support structure is modeled using beam elements to simulate deck flexibility. Nonlinear material properties and large deflections are also incorporated. A fluid domain outside the vessel is used for simulating the transient explosion acting on the external surface of the vessel, and an internal fluid domain is used for the fluids (liquid and gas) in the vessel. The time history of the explosion is applied at the inlet of the fluid domain upstream of the vessel. The explosion, advancing rapidly through the domain, results in high transient drag loading on the vessel, causing it to move. The movement of the vessel causes the internal liquid to undergo sloshing, generating additional loads on the shell of the vessel.

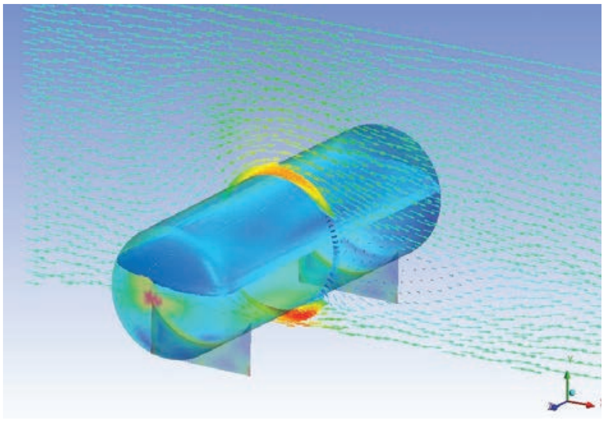
Grantec engineers leverage the ANSYS HPC (high-performance computing) multi-processor option to significantly speed up the computationally demanding analysis. The company migrated from two HPC licenses to the HPC Pack license, which resulted in a four-times speed improvement for multiphysics simulations on an HP Z820 workstation. The team is looking at ways to further improve turn-around on multiphysics simulations — such as using GPU capabilities, additional hardware and other methods.

The results of the research conducted will be used to assess and guide future standards requirements.

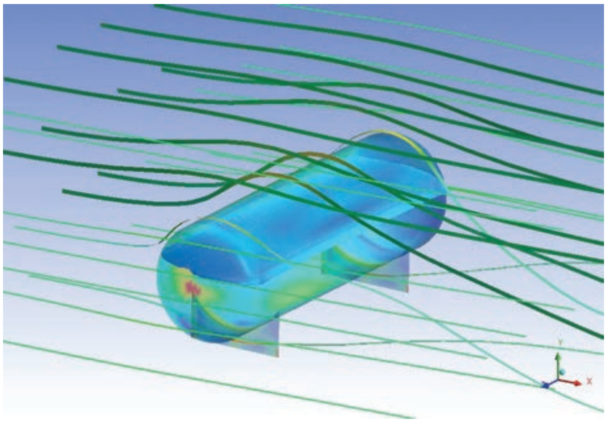
International cooperation between standards-making bodies helps to substantially improve offshore platform standards in the areas of structural integrity, control and mitigation of accidents, and protection of safety-critical systems. Simulation enables engineers to understand and diagnose many potential accidents and evaluate the effects of possible design standards in protecting human life and the environment.



Simulation of accidental ship collision with wellhead platform



Fluid–structure interaction simulates an explosion that induces load on pressure vessel. Image shows instantaneous velocity vectors of air moving as a result of explosion.



Time slice shows streamlines of explosion flow over pressure vessel.