SIMULATION TAKES WHIRLPOOL IN A COOLER DIRECTION

PAGE 11

COMPREHENSIVE MULTIPHYSICS
PAGE 4

ENTERPRISE-WIDE CAE
PAGE 8

ROTATING MACHINERY SPOTLIGHT
PAGE 15
Control complexity and complete projects twice as fast

When your applications all run in a single, powerful data environment designed specifically for engineering—projects can move from design to simulation, then on to completion, with ease. That's why we've built our storage and server solutions—including innovative Altix® XE servers based on the Quad-Core Intel® Xeon® 5300 processor—to work together and control complexities, even in the most challenging engineering environments.

Learn more at sgi.com/go/connectansys
A Matter of Survival

Simulation and statistical methods help manufacturers achieve high product quality now essential in competing on the world market.

Products designed using sound principles often fail once they are built and in use. That’s one of the frustrating puzzles of engineering: defective products based on designs that passed quality checks, engineering analysis and prototype testing with flying colors. In many cases, such failures are caused by the unforeseen interaction of multiple variables in production, materials, shipping and customer use — and manufacturers pay a steep price for these failures.

The key to evaluating these kinds of interactions is an approach called Design of Experiments (DOE), in which numerous random analyses are run on different combinations of changing variables. Probabilistic and statistical methods compare all the different results and study the sensitivity of product behavior to these variations. The tools are at the heart of Design for Six Sigma (DFSS) methods in arriving at optimal near-defect-free “robust” designs that work properly — even in the face of wide variations of product parameters.

For years, teams of statisticians, analysts, designers and experts have had to spend months plowing through thousands of simulations and mountains of data for such studies. Consequently, the DOE approaches were used mostly by large companies with hefty resources. The emergence of specialized technology such as ANSYS DesignXplorer software changed all this with automated features for quickly and easily performing many of these repetitive tasks, often completing projects involving 10,000 or more parametric analyses in a matter of hours. These capabilities enable design engineers to apply the same Six Sigma quality principles that, for years, have been such an important focus in manufacturing operations and a topic of strong interest among the ranks of corporate management.

Functionality for such processes is outlined in Pierre Thieffry’s article “Parametric Design Analysis for Evaluating a Range of Variables,” which describes tools for assessing the influence of all relative parameters on design objectives and system performance.

The ramifications are profound and potentially far-reaching for a broad range of manufacturing companies, including mid-sized and even small job shops, where engineers can use these methods to achieve high product quality — which is not a luxury anymore but increasingly a matter of survival in the competitive world market.

John Krouse, Senior Editor and Industry Analyst
Spotlight on Engineering Simulation for Rotating Machinery

15 As the World Turns
World conditions and increased competition challenge rotating machinery designers to deliver higher levels of performance, efficiency and reliability — faster than ever before.

17 Hot Streaks and Deformation
Software tools from ANSYS improve durability and reduce emissions in gas turbines by helping to reduce creep in combustion liners.

20 Innovative Diagnosis for Instability in Turbomachinery
Simulation helps to predict subsynchronous vibrations and rotordynamic stability for centrifugal compressors.

23 High-Speed Product Design
Integrated software facilitates design and development of expansion turbines to avoid failure.

25 Runners Experience Longer Life
Fracture mechanics helps ensure longevity of propeller-type runners in hydropower plants.

Spotlight on Engineering Simulation for Rotating Machinery

4 MULTIPHYSICS
Multiphysics in Action
Powerful coupled-physics simulation tools solve demanding applications in a wide range of industries.

8 THOUGHT LEADERS
CAE on the Offensive
A leading Australian aerospace and defense company, Tenix Defence Pty Limited, reports on computer-aided engineering software trends.

11 CONSUMER PRODUCTS
Keeping Cool While Cutting Costs
Simulation helps keep temperatures and costs down while optimizing refrigerator design.

SIMULATION @ WORK

27 MATERIALS
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.
28 MATERIALS
Savings from Submerged Combustion Melting
Simulation helps glass manufacturers understand complex phenomena in next-generation melter technology.

30 HEALTHCARE
Breathing Easily
Simulation of airflow in human noses can become a useful rhinosurgery planning tool.

3 HEALTHCARE
Ins and Outs of Inhalers
Simulation helps optimize the performance of a dry powder inhaler for drug delivery.

36 OIL AND GAS
Supporting the Oil and Gas Industry
Longevity and safety of drilling derricks and substructures are increased through stress analysis.

DEPARTMENTS

38 PARTNER
Integrated Analysis Achieves State-of-the-Art Workflow
A collaborative process and better tools help Modine engineers leverage the virtual environment to meet emission standard design changes.

40 PARTNER
From CAD to CAE
FLUENT software now offers support for Autodesk Inventor.

41 TIPS AND TRICKS
Analyzing Buckling in ANSYS Workbench Simulation
Simulation shows how parts catastrophically deform under compressive loads that exceed the structure's material strength.

44 ANALYSIS TOOLS
Parametric Design Analysis for Evaluating a Range of Variables
Tools help to study engineering trade-offs in Simulation Driven Product Development.
Multiphysics in Action

Powerful coupled-physics simulation tools solve demanding applications in a wide range of industries.

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

In an expanding range of applications, engineers must be able to accurately predict how complex products will behave in real-world environments in which multiple types of coupled physics interact. Multiphysics simulation is becoming crucial in the development processes for a rapidly growing number of companies. It has the potential to influence engineering simulation efforts in coming years, as more and more companies recognize the strategic value of the technology.

The increased demand for multiphysics simulation is occurring in many different industries as companies strive to maintain a competitive edge. In the electronics industry, high current densities in microchip circuits create large heat loads that need to be dissipated. In the automotive industry, airflow over exterior components, such as side-view mirrors, can create unwanted noise and vibration. In the biomedical industry, understanding how blood flows through stent grafts can be used to improve surgical procedures. In these and a growing number of other applications, multiphysics simulation is rapidly becoming a competitive necessity by allowing engineers and designers to closely evaluate their designs under real-world operating conditions.

Advanced Technology

Multiphysics simulation has been part of the core technology from ANSYS for several decades. From the early versions of the software that included thermal–stress calculations to the recent development of complex thermo-electric–fluidic calculations, ANSYS solver technology has continued to advance the development of state-of-the-art multiphysics solution capabilities.

The functionality of multiphysics technology from ANSYS is unparalleled, with no other solution provider able to match the long development history and the technical
depth of solution capabilities. Indeed, companies needing true “industrial strength” multiphysics capabilities continue to rely on the complete repertoire of industry-leading functionality from ANSYS.

The current version of ANSYS Multiphysics software has two proven solution techniques for solving coupled-physics problems — directly coupled-field elements and the ANSYS Multi-field solver. These approaches provide flexible simulation methods built on proven solver technology to solve a broad range of complex coupled-field problems, such as induction heating, electrostatic actuation, Joule heating and fluid structure interaction (FSI).

**Directly Coupled-Field Elements**

Directly coupled-field elements allow users to solve coupled-physics problems by employing a single finite element model with the appropriate coupled-physics options set within the element itself. ANSYS coupled-field elements account for coupled physics by calculating the appropriate mathematical terms that include the interaction between the different physics disciplines. In this way, coupled-field element solutions simplify the modeling of a multiphysics problem by allowing users to create, solve and post-process a single analysis model for a comprehensive array of multiphysics problems.

ANSYS coupled-field elements encompass a wide variety of multiphysics analyses including thermal-structural coupling, piezoelectricity, piezoresistivity, the piezocaloric effect, the Coriolis effect (the apparent deflection of moving objects from a straight path when they are viewed from a rotating frame of reference), electroelasticity, thermal-electric coupling and thermal-electric-structural coupling. The broad range of capabilities provided by these elements is essential for the design of many products, such as electronic components, micro-electro-mechanical systems (MEMS), transducers, piezoelectric gyroscopes, accelerometers and thermoelectric coolers.

In one such application, ANSYS coupled-field elements were used to evaluate the performance of a vibrating silicon ring gyroscope, a particular type of angular velocity sensor commonly used in automotive braking systems and vehicle stability control systems. The gyroscope is a solid-state device comprising a micromachined silicon ring suspended by surrounding spokes. The ring is excited into a primary mode of vibration and rotated while vibrating, setting up a secondary mode of vibration generated from the Coriolis effect. The angular velocity of the sensor is then detected by sensing the secondary mode of vibration, which is proportional to angular velocity.

One challenge in the design of silicon ring gyroscopes is minimizing energy loss for better sensor performance and lower power consumption. One of the most important
energy-loss characteristics to be evaluated in the development of the device is thermoelastic damping arising from the irreversible heat flow across the temperature gradients induced by the strain field. This effect is characterized by a strong coupling between the structural and thermal fields, accurately represented using matrix coupling in ANSYS Multiphysics software. In this way, the software enabled engineers to minimize energy loss in the gyroscope by evaluating complex mode shapes and the harmonic response of the silicon ring gyroscope while accounting for thermoelastic damping.

**Multi-Field Solver**

The ANSYS Multi-field solver solves a wide variety of coupled-physics problems by employing implicit sequential coupling. Examples include thermal–structural coupling, thermal–electric–magnetic coupling, electromagnetic–structural coupling and fluid structure interaction (FSI).

With sequential coupling, each physics discipline is solved sequentially, and results are passed as loads from one physics discipline to another with convergence between the individual physics disciplines obtained at each point during the solution. This robust convergence behavior of implicit coupling ensures accuracy and minimizes the engineering time needed to achieve valid simulation results.

Since two or more single-physics models are used within the ANSYS Multi-field solver, results can be passed across a dissimilar mesh interface between the physics disciplines. This is a subtle but very important consideration, since a dissimilar mesh interface allows a user to optimize the mesh for each individual physics discipline. For example, in a fluid structure interaction problem, the meshing requirements for the fluid are often different from those for the structure. A dissimilar mesh interface also allows independent users to set up their specific physics disciplines, which in turn allows for closer collaboration between physics experts.

Using such an approach, the ANSYS Multi-field solver was used to evaluate the switching speed of a digital micromirror, a commercially successful MEMs device used as the basis of Digital Light Processing (DLP) technology. In a DLP projector, a projected image is created by an array of several hundred thousand digital micromirrors that each alternate rapidly between on and off states, projecting light from the projector into a lens that focuses the pixels on the screen. Held in place by thin tethers, the tiny aluminum mirrors are repositioned during each cycle using electrostatic forces. By sequentially coupling electrostatics and structural deformation in this complex problem, ANSYS Multiphysics software was instrumental in evaluating the positioning of the digital micromirror as well as the switching speed of the system.

Another common application of the ANSYS Multi-field solver is fluid structure interaction, which occurs when a fluid interacts with a solid structure causing deformation in the structure and thus altering the flow of the fluid itself. An FSI solution is required for many industrial applications, such as the aerodynamic flutter of airplane wings, transient wind loads on buildings, and biomedical flows involving compliant blood vessels and valves.

For cases such as these, both the structural and fluid solutions must be run concurrently with loads transferred between the two solvers. ANSYS Multiphysics software provides a unique
Implicit coupling for fluid structure interaction includes iterative analyses of fluid forces that may cause structural reactions, which in turn may alter fluid flow.

Another unique capability of the ANSYS Multi-field solver is the ability to include coupled-field elements within an FSI solution. In one project, coupled-field elements were used in conjunction with the ANSYS Multi-field solver to evaluate the performance of a piezoelectric fan in which the motion of the fan blade is generated by applying a voltage to a piezoelectric material. Often used for spot cooling of electronic components, piezoelectric fans typically consume less energy than conventional fans and do not produce electro-magnetic noise that can interfere with computer circuits.

In this device, the fan blade is driven at resonance using a standard AC excitation of 120 volts applied at 60 Hz. ANSYS Multiphysics software was used to optimize the size of the fan blade to produce a resonant frequency of the device exactly at 60 Hz and to evaluate the motion of the fan blade and subsequent air flow. This solution required piezoelectricity to be coupled to computational fluid dynamics, which is a unique capability of ANSYS Multiphysics software.

**Multiphysics as a Strategic Tool**

By using such capabilities, engineers can now perform multiphysics analysis as a routine part of product development. In this way, coupled-physics simulation takes on strategic value in the development of virtual prototypes by accounting for all of the relevant physical phenomena that influence designs.

ANSYS continues to lead the industry in the development of multiphysics solutions that provide the high-fidelity simulations required to meet the challenges of today’s demanding product development requirements.

**ANSYS Multiphysics software was used in optimizing a piezoelectric fan blade geometry to produce a resonant frequency exactly at 60 Hz and in evaluating the motion of the fan blade and resulting air flow.**
CAE on the Offensive

A leading Australian aerospace and defense company, Tenix Defence Pty Limited, reports on computer-aided engineering software trends.

By Peter Wilson, Engineering Manager, Electronic Systems Division, Tenix Defence, Sydney, Australia
With comments from Fabian Ravalico, Engineering Manager, Land Division, Tenix Defence
Kerry Thurstans, Engineering Manager, Aerospace Division, Tenix Defence
Saeed Roshan-Zamir, Structural Engineering Manager, Tenix Marine Division, Tenix Defence

From a naval shipbuilding business in 1997, Tenix Defence Pty Limited has grown to service most areas of the defense industry; it also has established businesses working in aviation, parking and traffic infringement management, commercializing innovative technology and providing engineering services for utilities including water, sewerage, gas and electricity. Based in Sydney, Australia, Tenix operates in all mainland Australian states and territories, New Zealand, the South Pacific and Southeast Asia. It is also an active partner in high-technology ventures with United States and European firms. ANSYS Advantage magazine interviewed Peter Wilson, Engineering Manager of Tenix Defence, Electronic Systems Division, and his colleagues about computer-aided engineering (CAE) software trends at Tenix.

Note: Tenix was sold in late January 2008 to BAE Systems, which is Europe’s largest defense company. The acquisition makes BAE the biggest supplier of equipment to Australia’s armed forces.

Q: Who is Tenix and what is its role in the defense industry?
A: Tenix is one of the largest independent defense contractors and integrators in both Australia and Southeast Asia. There are four major defense businesses: Aerospace, Land, Marine and Electronic Systems. We have other interests in engineering systems, such as traffic cameras and domestic utilities including waste water treatment and electricity generation.

The company emerged from the industrial construction company Transfield, which was formed in the 1950s. We see our core competence as the ability to be the smartest defense integrators in our segment, and we seek to understand our customers’ individual niche needs. We then work with original equipment manufacturers to deliver best-in-class flexible, customized designs fit for purpose.

Tenix works extensively for the Australian government and other Southeast Asian and Australasian countries providing defense engineering solutions. We currently are building and delivering seven vessels for the Royal New Zealand Navy tailored to their unique systems integration needs. In addition, we work with major defense contractors, such as Lockheed Martin, L3 and Northrop-Grumman, to customize their off-the-shelf defense equipment.

Q: What sort of technical systems integration do you typically perform, and how do computer simulation and CAE feature in your processes?
A: This varies significantly among our defense divisions, and it is best answered on a case-by-case basis, as follows.

By Peter Wilson, Engineering Manager, Electronic Systems Division, Tenix Defence, Sydney, Australia
With comments from Fabian Ravalico, Engineering Manager, Land Division, Tenix Defence
Kerry Thurstans, Engineering Manager, Aerospace Division, Tenix Defence
Saeed Roshan-Zamir, Structural Engineering Manager, Tenix Marine Division, Tenix Defence
Electronic Systems Division: Peter Wilson, Engineering Manager

The Tenix Electronics Systems Division (ESD) conducts many research and development-type projects. As such, we have a high focus on new design rather than the evolution of existing products and systems. CAE plays an important role in this line of work, as it allows our engineers to communicate and deliver designs that, due to their developmental nature, can change rapidly in scope. Our key technology areas, including electronic warfare, high-power lasers and electro-optics, see simulation of fluid flow and thermal interactions as a high priority, with structural analysis and modal response being the next most used simulation capabilities.

The tools from ANSYS have already demonstrated benefits: increased confidence in design solutions leading to less conservative designs and reduced solution iterations in the high-technology defense sector in which ESD conducts its business. All of ESD’s mechanical and aeronautical engineers now have been trained to use the ANSYS tools. We are part of a powerful community of users among the various divisions of Tenix that is able to share its experience, ideas and workload.

Land Division: Fabian Ravalico, Engineering Manager

Land Division’s core business is based on the development, modification, upgrading and through-life support of military and commercial armored vehicles. Product development traditionally involves physical prototyping followed by testing and introduction of improvements in an iterative manner until the design is mature, verified and validated. The use of modeling and simulation of design concepts via CAE tools leads to accelerated product development lead times; it also reduces the amount of iteration required to reach a mature design. It allows all concepts and design solutions to be considered and assessed for validity in a relatively compressed timescale, enabling progression to a hardware solution (prototype) with high confidence of success. Modeling and simulation are very often more cost-effective than prototyping.

Using Tenix’s new suite of tools from ANSYS, Land Division’s most commonly explored analysis domains are linear analysis, nonlinear analysis, dynamic analysis, crash analysis and blast analysis. Our new challenge is to provide the necessary staff training to capitalize fully on the new tools available.

Marine Division: Saeed Roshan-Zamir, Structural Engineering Manager

We use software from ANSYS extensively to design and assess a large variety of marine structures. This principally involves the solution of load cases, such as shock and airblast, fatigue, vibration and operational loadings at sea. The most notable example is the structural design and analysis of new masts on the ANZAC-class frigates for the Anti-Ship Missile Defence (ASMD) project, which involves the integration of a precision targeting, tracking and illumination phased array radar system. Often, we utilize transient simulations to model and capture the response of the structure subject to shock pulse accelerations and blast pressures; in addition, we commonly require nonlinear boundary conditions and modal results to capture the natural frequencies of the structure. We employ CAE for our tasks because we require detailed and accurate results in order to effectively optimize our designs. A high percentage of our staff needs to be capable of using the programs effectively — so it’s important for us that the programs can be learned quickly and successfully. We have found software from ANSYS to be highly capable of meeting our needs.

Aerospace Division: Kerry Thurstans, Engineering Manager

Tenix’s Aerospace Division undertakes work in the Australian defense aerospace environment as the lead (or prime) company for the systems integration of avionics, communications, electro-optics, electronic warfare (EW) self protection, and other advanced commercial and military systems. During the design and testing phases of many of our projects, we make extensive use of computer-aided engineering to complement classical analysis and physical testing.
Using a company-wide common toolset, as we have implemented with ANSYS, provides a common training environment, encourages intra-company collaboration and facilitates knowledge transfer.

Q: How do you see your CAE usage evolving in the future?

A: The company’s recent decision to choose simulation packages from ANSYS as enterprise-wide CAE tools means that we can improve our productivity and design flexibility like never before for all of our divisions. We selected ANSYS because of the proven performance of the toolset across our broad range of CAE requirements along with the collaborative approach adopted by ANSYS and their Australian agent. We also like the way that we can access a wide range of CAE tools in the one common ANSYS environment, all during the same work session.

One of the benefits of using software from ANSYS in all Tenix divisions is that it allows us to increase and improve our in-house CAE capabilities. We want our existing engineers to use CAE widely. In addition, we intend to hire more engineers to use these exciting new design tools, and the concept of a one-stop design house will certainly be attractive to them.

Going forward, Tenix is looking to increase its capability in analyzing fluid and structure simultaneously using fluid structure interaction simulations. We certainly have the need to extrapolate existing simulation models with confidence and develop a well-validated set of CAE capabilities.

Q: What does Tenix see as the biggest defense sector CAE challenges for the foreseeable future?

A: Personally, I want Tenix to have a strong integration of the engineering capabilities within all four defense divisions with a close coupling of our technical know-how and expertise. I see CAE as the latest part of that revolution. Already, we have integrated our systems engineering requirements management tools, and product lifecycle management (PLM) is next on our list.

By getting enterprise-wide usage of these same tools, I believe it will lead Tenix to significant productivity gains and efficiency savings as we develop a virtual community of internal users. Over the years, the separate divisions at Tenix have been geographically dispersed across Australia, and they have developed their own expertise. We need to cross-fertilize this CAE knowledge with our deployment of a common integrated software toolset from ANSYS.
Global competition in the appliance industry is placing ever-increasing pressure on manufacturers to decrease costs while maintaining high quality. Refrigeration products are especially competitive, and as raw material costs for these appliances continue to rise, manufacturers are forced to take a close look at their product designs.

With worldwide industry revenue in 2006 reported at $6.2 billion and a sluggish market to contend with, the stakes are high. In an effort to find a competitive edge, Whirlpool Corporation, one of the market leaders in the consumer and commercial refrigeration market, has turned to software from ANSYS. In the past, cost-reduction projects at Whirlpool typically have required that engineers build and test several prototypes, with results then compared with current production cabinets. This trial-and-error approach is costly and time-consuming, leading to only incremental changes. Recently, Whirlpool has enjoyed more substantial benefits...
from an approach that utilizes leading-edge simulation combined with experimental tools to assess complex structural behavior.

When Whirlpool recently looked to cut costs associated with producing a three-year-old, 450-liter double-door refrigerator, the model was required to meet the existing specific cabinet deflection and door drop limits of its current design as well as maintain adequate cabinet stiffness. Cabinet deflection and door drop occur when a fully loaded door is opened. When this occurs, the cabinet distorts and the door moves downward, eventually leading to cabinet deformity. Additionally, any redesign that changes cabinet stiffness could negatively impact insulating capabilities as well as aesthetics, which, in the end, can influence consumer-perceived quality.

A refrigerator cabinet is constructed of external sheet metal parts, polyurethane foam filling and internal plastic liners. It is very complicated to evaluate cabinet deflection and door drop since there is significant variation between products of the same model, with variations in both the manufacturing and testing procedures affecting door stiffness. To identify the most significant variables, engineers at Whirlpool used Design of Experiments (DOE) and sequential analysis. Following this process and the compilation of quantitative data describing structural behavior, the engineering team then utilized ANSYS Mechanical software to optimize the refrigerator design.

The goal of simulation and analysis was to evaluate the design factors that most significantly affect material costs and door drop. To begin, engineers constructed a finite element model (FEM) of the cabinet using solid and shell elements. (See technical sidebar for details.) All of the finite element analyses (FEAs) executed during this study were performed using ANSYS Parametric Design Language (APDL). The state or response variable used to control the cabinet stiffness was door drop under static load application. The cost function to be minimized was directly related to the mass of the cabinet sheet metal parts.

With the goal of reducing costs by cutting the amount of materials used in manufacturing, the analysis showed that reducing the thickness of the compressor plate, deck reinforcement, bottom deck and front rail (7.8 percent of total mass) would allow for a reduction in materials while having a minimal effect on door drop. The simulation/optimization further determined that the most

---

**Simulation Setup**

The refrigerators’ polyurethane foam, EPS mullion, hinges and levelers were modeled with solid tetrahedron elements (SOLID45). Other parts, basically composed of thin plates, were modeled with SHELL181 elements. For all the connections of clinch joints and screws, the element BEAM188 was utilized.

All of the material properties were considered as linear isotropic, and the input data was derived from laboratory tests and supplier technical specifications. For the polyurethane foam, the elastic properties were evaluated in a laboratory test device, and the elasticity modulus was included as a factor.

In order to adequately represent the real loading conditions, the masses of all cabinet parts were evaluated and included. In addition, the masses corresponding to cabinet ballast and loaded doors were calculated and were then assigned to each location in the model. The acceleration due to gravity was applied as a load boundary condition. At its bottom base, the cabinet was constrained, simulating actual operating conditions.

The door attachments and hinges were modeled with constraint equations, which allowed controlling translations and rotations to be correctly transferred from the doors to the cabinet. As a result, it was possible to impose door support only at the bottom position and release rotations from the hinge pins. In so doing, it was necessary to set two additional constraints at the upper liner of each door to eliminate rigid body motion.
sensitive factor affecting mass was the cabinet wrapper (or outer paneling), but that reducing the wrapper thickness resulted in an increase in door drop. The most significant factor driving door drop was the screw that connected the intermediary rail and cabinet front flange. In order to compensate for the increase in door drop that resulted from reducing the wrapper thickness, designers added two screw connectors, rather than one, between the wrapper front flanges and the intermediary rail, resulting in a 12 percent improvement in door drop. This design change also contributed to cabinet robustness and compensated for polyurethane foam stiffness loss (manufacturing process variation).

In the end, this optimization and the associated design changes resulted in the reduction of overall cabinet mass by 26 percent while maintaining door drop and cabinet displacement at reasonable levels, as defined by Whirlpool quality standards. On the bottom line, material costs were reduced by 15 percent per product, resulting in a cost savings for the company of $1.2 million per year.

By using ANSYS Mechanical software and Six Sigma tools, analysts at Whirlpool now have the ability to develop complex finite element cabinet models calibrated with real data, enabling the optimization of first-round physical prototypes. This procedure has resulted in a reduction of development time and manufacturing costs, helping Whirlpool meet increasingly competitive market requirements.

---

**Six Sigma and the Importance of Critical Thinking**

In this case study from Whirlpool, Six Sigma tools, including Design of Experiments (DOE) product testing, were used to assess manufacturing process and laboratory test sources of variation affecting cabinet structural behavior. Eventually, engineers gained enough knowledge to act on variation reduction in order to accomplish the finite element model (FEM) calibration.

With the FEM and calibration completed, the first optimization in ANSYS Mechanical software executed a sequential virtual DOE, with factors and levels selection based on Six Sigma tools. The goal was to evaluate the design factors that most significantly affected door drop and material costs.

At Whirlpool, Six Sigma initiatives are used to investigate the impact of sources of variation on key critical outputs of product and process, such as quality or performance, and the initiatives are often lauded for huge reductions in defects. A phrase often associated with Six Sigma philosophy is “Y = f(X)” (Y is a function of X.) This overly simplified equation reflects the observation that behavior in critical product or process performance characteristics (Y) is due to certain process factors or inputs (X). For example, the cabinet wrapper thickness (X) can potentially affect cabinet stiffness and door drop (Y). A crucial part of Six Sigma work is to define and measure variation in Y with the intent of discovering the cause and developing efficient, operational means to control, mitigate or reduce the variation.

In addition, critical thinking based on the scientific method is a very important skill and is used extensively to conduct industrial experiments associated with simulation, which then leads to design optimization. Critical thinking — the process of deduction and induction — implies that the investigator has the wherewithal to develop theories from the initial questions. This wherewithal comprises subject matter knowledge, experience, process knowledge and the ability to reflect critically and engage others. In fact, it has been shown that critical thinking is more important to improvement and development activities than training in a particular tool set.
More than 250,000 professionals worldwide are using Mathcad to perform, document, and share calculation and design work. The unique Mathcad visual format and easy-to-use whiteboard interface integrate standard mathematical notation, text, and graphs into a single worksheet—making Mathcad ideal for knowledge capture, calculation reuse, and engineering collaboration. Mathcad lets individuals work with updatable, interactive calculations, so users can capture the critical methods and values behind each of their engineering projects.

**Mathcad and ANSYS Integration:**

ANSYS users can enjoy added benefits with Mathcad – ANSYS integration:

- Allows Mathcad to act as an “add-in” within Workbench to manage parameters.
- Engineers can automatically exchange data and calculated values between Mathcad and the ANSYS Finite Element Analysis modeling solution, supporting dynamic, live updates to calculations.

Mathcad engineering calculations associated with the analysis model can dynamically drive multiple iterations in ANSYS Workbench.

**Mathcad Free Trial.** Check out these benefits and more with the Mathcad free 30-day trial: [PTC.com/go/mathcadfreetrial](http://PTC.com/go/mathcadfreetrial)

The Mathcad – ANSYS integration is a demonstration integration that you can download free. It is intended to be edited and customized to serve your own specific needs. As such, it is not officially supported by PTC.
As the World Turns

World conditions and increased competition challenge rotating machinery designers to deliver higher levels of performance, efficiency and reliability — faster than ever before.

By Brad Hutchinson, Vice President Industry Marketing, Aerospace and Turbomachinery, ANSYS, Inc.

Turbomachinery, or more broadly speaking rotating machinery, spans almost all industry sectors and in many plays a vital role. Rotating machines change the state of working fluids (pumps or compressors), convey or transport fluids (fans and pumps), extract energy (turbines) and create propulsion (propellers). Performance, efficiency, reliability and rapid delivery have always been important, but today’s world conditions intensify the pressures designers face.

Currently, the price of oil hovers near $100 per barrel. Concern for climate change is widespread, not only by the general public but also by legislators worldwide. Since rotating machines play a critical role in power generation and various forms of transportation — and, in some cases, are the limiting factor regarding cost, efficiency and emissions — it is natural that they are in the spotlight and the subject of intense analytical scrutiny.

In the energy industries, most of our power is produced by gas, steam and water turbines. Steam turbines extract power in nuclear and coal-fired power plants. Land-based gas turbines, similar to their aeroengine cousins, run on natural gas and, in some cases, oil. Although still relatively small in volume, wind turbine production has increased...
ROTATING MACHINERY: OVERVIEW

dramatically in recent years, with the largest machines being 6 megawatts (MW) in size and having rotors approaching 130 meters in diameter. The largest water turbines, such as those used in the Three Gorges Dam in China, have 10-meter diameter runners.

In the transportation industry, turbomachinery plays an equally important role. For air travel, the rotating machinery used on commercial aircraft is well known — the gas turbine aircraft engine. Its key rotating components include the fan, which can be seen when boarding the plane, as well as the compressor and the turbine. Since fuel is a major and often volatile cost for airlines (significantly impacting their profitability) and noise and emissions regulations are becoming increasingly stringent, there is a drive for cleaner, quieter and more fuel-efficient engines.

For transportation at sea and on the ground, diesel engines in ships, trucks and an increasing number of cars use turbochargers to improve their performance and efficiency. These engines also use electric-driven fans and pumps, which must be optimized, since available electrical power is limited. Efficiency of the automatic transmission torque converter — another rotating machinery component comprised of a pump, a stator and a turbine — is critical to vehicle fuel efficiency.

Turbomachinery plays an important role in other industries as well. Compressors and pumps are important to the chemical, process, and oil and gas industries, and are even key components in large industrial air conditioning systems. In the medical industry, heart pumps must be designed to be compact and to minimize blood damage.

Each rotating machine type has one or more key design challenges. Cooling due to high temperatures is problematic in gas turbines. Cavitation is an issue in pumps. Non-ideal gas behavior in steam turbines and refrigerant compressors must be considered. In aeroengine fans, noise is a challenge. There are space limitations when designing automotive fans. For hydraulic turbines, large-scale transient instabilities — also known as vortex ropes — may occur at off-design conditions. These examples indicate the diverse physics requiring consideration for accurate simulation of real machine behavior. In reality, these physical processes interact, and multiphysics analysis increasingly is becoming a requirement for high-fidelity simulations.

Several design, performance or production factors are common among turbomachine types. Reliability and safety require accurate prediction of steady and transient thermal, aerodynamic and structural loads for stress and fatigue life predictions. However, excessive safety and strength features are likely to make the machine too expensive or too heavy; these features also can preclude other competing requirements such as efficiency.

Operational cost and emission issues have recently intensified the pressure to produce efficient machines. Time-to-market and cost pressures resulting from competition have underscored the need to “get it right the first time.” These factors, in turn, demand that simulation software provides solutions of ever-increasing resolution and accuracy. Software, when employed in Simulation Driven Product Development (SDPD), helps designers resolve these and other challenges and is a key enabler for reduced-cost, first-to-market rotating machinery development.

References

A low-emissions combustion liner is a critical system component for gas turbines. The combustion air in a gas turbine enters through holes in the combustion chamber liner and flows along the liner to keep it cool. Liners are designed to improve durability and cooling while minimizing the flow variation from liner to liner within the same engine. Reducing variation can decrease exhaust temperature spreads, engine hot streaks and emissions. By combining combustion, flow and structural modeling using tools from ANSYS, Power Systems Manufacturing (PSM) in Florida, U.S.A., designs virtual prototypes and avoids expensive physical testing until the very end of the design cycle.

Land-based gas turbines are often used in so-called “peaking” units, when the demand of the electric grid overwhelms the base-load capacity (most often handled by coal or nuclear units). This means that, in order to meet the grid demands but not to exceed them, gas turbine generators often run at partial or varying load. Modern can-annular F Class combustion systems, such as PSM’s Flamesheet, are typically composed of several fuel stages designed to enable the gas turbine to ramp up or down in load during startup and shutdown, or to allow it to run at partial loads. PSM’s Flamesheet combustor is composed of three separately fueled equiangular circumferential main stages and a pilot stage at the center. The pilot stage is typically operated at low loads, and the main stages are brought online one stage at a time as the turbine ramps up. The combustion liner has only a circumferentially balanced flame during the pilot stage and full-load operation. At part-load conditions, when only one or two of the main stages are on, the flame is located on one-third or two-thirds of the liner inner surface, thereby producing a hot streak along the liner wall.

The high thermal variation and asymmetry on the metal surface of the combustion liners resulting from these hot streaks cause thermally induced stresses. Stresses can lead to thermo-mechanical fatigue that accrues on a cyclic basis and results in low-cycle fatigue failure. If the thermally induced stresses are lower than the yield limit of the material, the cylindrical liners may still deform inelastically due to creep relaxation over time. Liner deformation affects the structural integrity of the combustor as well as the circumferential fuel–air mixture distribution coming out of the premixer, which, in turn, can have a negative impact on the unit’s emissions.

Engineers at PSM used a finite element (FE) technique and ANSYS Mechanical software to design a liner to mitigate such creep ovalization effects. The analysis predicted creep deformation over time of Haynes 230 alloy liner material using inputs from creep specimen testing. The engineers validated these results against field test data of PSM’s Flamesheet combustion liners carried out in a fully-instrumented combustion system of a Siemens Westinghouse SW501F unit at Calpine Corporation’s South Point Plant in Arizona, U.S.A. Similar inspection of 7FA DLN 2.6 combustion system liners also showed signs of creep ovalization.
Three-dimensional FLUENT computational fluid dynamics (CFD) simulations provided thermal boundary conditions for the FE analysis under part-load operating conditions. Thermal gradients were highest at the premixer exit of the liner’s unsupported forward end, causing it to deform freely and assume a thermally distorted shape.

Next, the engineering team performed a linear elastic analysis on a full 3-D ANSYS Mechanical model that was constrained at the liner lugs to simulate the configuration of the installed engine. They mapped thermal profiles and pressure loads onto the model that were consistent with measured data for a two-stage part-load condition. They then isolated the stresses caused by the circumferential temperature streak by subtracting the stress due to the radial temperature change from the total outer diameter to inner diameter hoop stress data at the liner premixer axial location. The residual deformation due to the circumferential temperature gradient was isolated in a similar fashion and compared against post-operating inspection data after approximately 20 hours of operation. The linear elastic strains from the 3-D model were used to evaluate the creep strain due to the favorable comparison with experimental data.

Using liner drop measurements separated by several days of operation, PSM engineers found radial ovalization to be approximately 0.78 percent; this indicated creep relaxation. Furthermore, the creep data for the Haynes 230 example revealed that such creep deformations occurred within the first few hours of part-load operation. The linear elastic stresses were within the elastic yield limit of the liner material. The linear elastic strains associated with these stresses created a strain control environment in which the liner thermal stresses creep relax to the Haynes 230 creep strength capability. Since there are no constant stresses and thermal stresses are within the elastic limit, engineers expect the liners to achieve a permanent set based on the ovalized linear elastic shape and to maintain that thermal shape without further deformation.

Estimating the amount of creep relaxation under strain control (permanent set) in a combustion liner required the use of an FE analysis. Complexity was introduced by 3-D variations in temperature, stresses, creep strain and strain rate across the liner. Specimen lab testing for tensile creep data revealed that the Haynes 230 example exhibited negligible primary creep.

PSM engineers estimated the material constants in Excel through best-fit coefficients to the experimental data. They constructed a complete Flamesheet liner model using 3-D ANSYS SOLID186 element types to simulate the liner’s secondary creep behavior. They mapped temperature and external static pressure loads from the FLUENT results onto the model. The implicit creep routine in ANSYS Mechanical software was invoked by using the strain “rate = 1” option. The ANSYS Mechanical model simulated 25 hours of run time followed by a shutdown, then a restart and continuation up to 1,000 hours of operation followed by a shutdown.

The simulation results showed that the liner mixer exit accumulated approximately 0.55 percent (maximum radial deformation minus minimum radial ovalization) of radial ovalization after 25 hours of part-load operation. After 1,000 hours of operation, the mixer exit accumulated approximately 0.73 percent of diametric ovalization, which compares well with the 0.98 percent of diametric ovalization observed by coordinate measuring machine (CMM) inspection performed on the liner mixer exit after testing. This result also confirmed that the liner creep strain rate decreased as the liner creep relaxed to the desired thermal shape. Although the ANSYS Mechanical creep analysis under-predicts the ovalization, over-firing during testing
may have caused the liner temperatures to be higher than those predicted by the thermal analysis, resulting in the mismatch.

Using the ANSYS Mechanical simulation to determine the effects of part-load operating hot streaks on the PSM Flamesheet combustor revealed creep relaxation of the liners under a strain-controlled environment. Numerical predictions of the residual ovalization compared well with inspection data on the liner mixer exit obtained from testing. PSM is currently using ANSYS Mechanical solutions for investigating new designs in order to mitigate such creep relaxations in Flamesheet combustion liners.

References
Innovative Diagnosis for Instability in Turbomachinery

Simulation helps to predict subsynchronous vibrations and rotordynamic stability for centrifugal compressors.

By J. Jeffrey Moore, Program Manager (Rotordynamics Program), Rotating Machinery & Measurement Technology Section, David L. Ransom, Senior Research Engineer, Rotating Machinery & Measurement Technology Section and Flavia Viana, Research Engineer, Fluid Dynamics & Multiphase Flow Section, Southwest Research Institute, Texas, U.S.A.

The energy industry — particularly the natural gas and hydrocarbon segments — depends on centrifugal compressors to produce, process, liquefy and transport many different gases. As the pressures in a compressor increase, the dynamic behavior at shaft and impeller seals, axial thrust balance pistons and impellers becomes more complex, with vibration ultimately becoming a concern.

There are two types of vibration of concern in industrial compressors: synchronous vibration and subsynchronous vibration. Synchronous, or running-speed vibrations, normally are excited by residual unbalance resulting from small imperfections in the manufacturing and assembly processes. The second and more troubling type of vibration, subsynchronous vibration, occurs when non-conservative whirling forces (cross couplings) act to excite a lateral natural frequency, which occurs in cases in which these fall below running speed. The excitation forces generated at seals and impellers have components that act at right angles to the displacement vector. Cross-coupling effects tend to sustain whirling motion at a subsynchronous natural frequency when insufficient damping is present. The whirling motion is referred to as self-excited rotordynamic instability, and it can lead to serious damage if not properly controlled. Researchers at Southwest Research Institute (SwRI), a nonprofit applied engineering research and development organization headquartered in the United States, have examined the possibility that computational fluid dynamics (CFD) can be used to study subsynchronous vibrations and rotordynamic instability for centrifugal compressors.

A CFD analysis by Moore and Palazzolo[1] used a grid perturbation method (GPM) approach with a 3-D structured computational mesh to demonstrate how cross-coupled stiffness for liquid (incompressible) pump impellers could be determined. Gas forces in a compressible fluid tend to be smaller, more difficult to predict and more difficult to model than the liquid forces that were predicted by Moore and Palazzolo. Furthermore, the energy equation and an equation of state are required to completely describe the fluid flow. A lack of accurately predicted operational specifications for compressor designs can result in unexpected, dangerous, and damaging instabilities and subsynchronous vibrations, making the identification of accurate analysis methodologies essential to the industry.

**Description of Computational Model**

A compressor manufacturer provided SwRI with the complete geometry, process and rotordynamic information for a centrifugal compressor that previously had experienced subsynchronous vibrations. The centrifugal compressor was equipped with only four out of 10 stages and had undergone development testing a number of years prior. While testing at a speed of 21,500 rpm and a discharge pressure of 2,300 psi using nitrogen as the testing medium, the compressor encountered classical rotordynamic instability. The frequency corresponded to the first natural frequency of the rotor. A second instability was reached while operating at 23,000 rpm. Engineers at SwRI identified this particular compressor as suitable for a case study because the impeller aerodynamic cross coupling was the dominant effect on the machine’s stability. Because the exact conditions at which the compressor went unstable were available from test records, the CFD could be tested under the same conditions.

![Image of compressor mesh with boundary condition surfaces and sliding interfaces](image-url)
To generate a complete CFD model of this impeller — including both the primary and the secondary flow around the impeller/diffuser, shroud, back face and seals — the engineering team used ANSYS CFX technology. The shroud was displaced in the radial direction. Although the physical problem appeared to be inherently time dependent, a transient CFD solution of this problem was not required if a simple reference frame transformation was performed. Since the shroud region was solved in the whirling frame of reference, while the primary impeller passage was always solved in the rotating frame, a sliding interface was employed. Researchers chose the frozen rotor sliding interface approach exclusively, so as not to artificially constrain the circumferential pressure field. The rotordynamic influence of the labyrinth seal was modeled in a rotordynamics model using a traditional bulk flow seal code.

Researchers evaluated the rotordynamic force coefficients of the impeller by determining the impedance at a minimum of three precessional frequencies. For improved accuracy over a wide range of precessional frequencies, more than three would need to be calculated and a least-squares curve fit to the linear second-order model was performed. The coefficients of the curve fit would yield the impeller’s stiffness, damping and mass force coefficients.

Validation of Results

In this stage of the project, the team validated the results from the CFD and rotordynamic analyses using real-world data. They used two separate verification methods for the SwRI CFD model for impeller force work performance. In the first test case, researchers essentially reproduced the results of Moore and Palazzolo[1], though they used an unstructured mesh. This case demonstrated good correlation to previous predictions and experiment, validating the use of an unstructured grid. In the second verification, the team required a comparison of CFD-based stability predictions for a centrifugal gas compressor against measured sub-synchronous vibrations on the test compressor.

Overall, SwRI engineers found the CFD results to be in reasonable agreement with the performance data. The flow field in the secondary passage was highly recirculating. Using a second-order curve fit, the full set of force coefficients was computed. Since the team performed a CFD analysis on only stages one and three, normalized parameters were used to calculate the coefficients for stages two and four. These derived force coefficients were close to the CFD values, validating the method used. Researchers also performed a rotordynamic analysis to analytically determine total dynamic behavior of the rotor at high rotational speeds and the stability of the compressor rotor — including the effects of rotor flexibility, bearing stiffness and damping, eye seal stiffness and damping, balance piston stiffness and damping, and aerodynamic excitation.

Engineers analyzed two compressor instability cases: instability point one (21,500 rpm) and instability point two (23,000 rpm). Even though the speed increased for point two, the discharge pressure at the point of instability was approximately the same. Therefore, the predicted rotordynamic stability varied only slightly between the two conditions. For each case, the aerodynamic cross coupling was varied from 0 to about 25,000 lbf/in to define the slope of the stability curve. The point at which the lines intercepted the vertical axis represented the system stability without the effects of aerodynamic cross coupling. The point at which the lines crossed the horizontal axis was the stability threshold, beyond which the machine was predicted to be unstable. These two lines provided insight into the sensitivity of the rotordynamic stability as a function of aerodynamic cross coupling (Figure 6).

The CFD results showed much-improved agreement in overall magnitude in comparison to the application programming interface API and SwRI methods, which are empirically based equations that have been used in the industry for many years. CFD predicts similar levels of cross coupling for the two instability points, while the empirically based methods do not.

The SwRI team then performed a parametric study to determine the various parameters known to affect the flow field inside the impeller and their effect on rotordynamic forces. Based on this
study, they developed a new formula to describe impeller cross coupling. The formula stated that the cross coupling was proportional to the dynamic pressure and the axial length of the impeller, and inversely proportional to relative flow due to the change in the exit flow angle of the impeller, as shown below.

\[
K_{xy} = \frac{C_{xy} \rho_{dis} U^2 L_{shr}}{Q/Q_{design}}
\]

in which

- \(K_{xy}\) = cross-coupled stiffness of impeller (lb/in) [N/m]
- \(C_{xy}\) = constant for a given impeller design
- \(\rho_{dis}\) = discharge density (lbm/ft^3) [kg/m^3]
- \(U\) = impeller tip speed (ft/s) [m/s]
- \(L_{shr}\) = axial length of shroud from impeller eye seal to impeller tip (in) [m]
- \(Q/Q_{design}\) = flow relative to design flow

Subsequent studies have indicated that \(C_{xy}\) can vary for different impeller geometries, and it is typically in the range of 4 to 7.5.

As demonstrated in this study, the SwRI engineering team was the first to develop analytical methods capable of analyzing the rotordynamic forces on a centrifugal compressor impeller using CFD. The results compared favorably when predicting the instability of a full-scale compressor. Based on this result, the team concluded that the majority of the destabilizing force of a centrifugal impeller arises from the shroud passage, not the impeller-to-diffuser interaction. These results are described in more detail in Moore, Ransom and Viana[2].

**References**


---

### Table Stages 1 and 3

<table>
<thead>
<tr>
<th>Method</th>
<th>Instability Pt 1</th>
<th>Instability Pt 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>SwRI</td>
<td>18,453 lbf/in</td>
<td>23,441 lbf/in</td>
</tr>
<tr>
<td>API</td>
<td>13,848 lbf/in</td>
<td>17,301 lbf/in</td>
</tr>
<tr>
<td>CFD</td>
<td>12,098 lbf/in</td>
<td>11,908 lbf/in</td>
</tr>
</tbody>
</table>

---

![Figure 4. Summary of compressor performance for compressor stages 1 and 3](image1)

![Figure 5. Comparison of modally weighted aero cross-coupling values using the various prediction methods](image2)

![Figure 6. Stability curves for the compressor under analysis and various predicted values for which the compressor would become unstable (i.e., prediction values for where the curves would cross the x-axis of this plot; the CFD results provide the most accurate prediction).](image3)
The air separation industry relies on efficient and reliable turbomachinery to create the highest performance air separation plants possible. Key to this industry are expansion turbines — centrifugal or axial flow turbines that expand a high-pressure gas to reduce the gas temperature and produce work. The turbines are widely used for industrial applications that require fluid cooling or low temperature processing. More than 25 years ago, Praxair, Inc., of New York, U.S.A., a leading supplier of atmospheric, process and specialty gases, created an in-house turbomachinery group that specializes in cryogenic expansion turbines. For the last two decades, Praxair turbine design engineers have been using the suite of finite element analysis (FEA) products from ANSYS as their mechanical simulation software packages of choice.

Some of the key components of expansion turbines are radial inflow turbine impellers, which operate at very high rotational speeds. Impeller aerodynamic performance and reliability depend in part upon the impeller blade shape and thickness. In addition to steady-state centrifugal pressure and thermal loads, dynamic stresses arising from upstream flow nozzle pressure fields can cause impeller fatigue failure. The ability to accurately and quickly predict stress, deflection and the modal characteristics of an impeller allows Praxair’s turbomachinery designers to develop an impeller that provides maximum aerodynamic performance without sacrificing reliability.

Praxair engineers use the design analysis package BladePro-CF™ from Impact Technologies, together with mechanical simulation software from ANSYS, to perform steady-state stress analysis, modal analysis (natural frequency and mode shape), harmonic forced response analyses and fatigue life calculations. BladePro-CF is fully integrated with some FEA products from ANSYS (such as ANSYS.
Multiphysics, ANSYS Mechanical and ANSYS Structural licenses), making the coordinated use of products relatively simple.

In order to assess an impeller’s margin against fatigue failure, engineers use a combination of steady-state and dynamic stresses. The use of Campbell and interference diagrams, as well as animated mode shapes from BladePro-CF, allows engineers to visualize the potentially dangerous interactions of various impeller mode shapes and sources of excitation.

An analysis begins when Praxair engineers import basic impeller geometry data into BladePro-CF. They attach boundary conditions and select materials within BladePro-CF prior to creating the 3-D ANSYS model for simulation. Next, they apply pressure profiles, temperature profiles and the centrifugal load, and the mechanical software from ANSYS calculates the static stress throughout the impeller. Once the static stress analysis is complete, engineers examine plots of displacement, von Mises equivalent stress (for crack initiation) and maximum principal stresses (for crack propagation).

The use of a sound mesh topology and a high-density hexahedron-based mesh in the blades and shroud are crucial to accurate frequency prediction for the large number of modes of interest. For Praxair’s shrouded impellers, more than 100 modes of vibration are present, ranging from zero rotations per minute (rpm) to the highest frequency of interest. A very accurate representation of stiffness and mass is required to produce sufficiently accurate predictions. The BladePro-CF program pre-selects master degrees of freedom and then allows the Praxair engineers to modify these.

The FEA simulation then calculates the natural frequencies and mode shapes at either zero rpm or a defined speed that would include stress stiffening effects. The back-substitution files are saved for any subsequent harmonic forced response analysis. For radial inlet turbine impellers, it is important to identify diametral and circular mode shapes that are critical in assessing the likelihood of dangerous resonance conditions — as not all modes can be excited or are of equal importance.

Engineers next use dynamic forced harmonic response analysis to calculate the dynamic stresses. At this step, a Goodman diagram (available in BladePro-CF) is invaluable, as it provides a graphic display of the combinations of static and dynamic stresses for the entire impeller. The diagram allows a viewer to visually compare the combined stresses with the material’s allowable limits. Once the critical locations are identified, engineers utilize the local strain module of BladePro-CF to calculate the time to crack initiation for each location; this value combined with the duty cycle of the compressor is then used to predict fatigue life.

The Praxair engineering team was able to pinpoint a weakness in an older impeller that experienced a fatigue failure by looking at a Campbell diagram. In so doing, they easily determined that vibration-based failure could be avoided by changing the number of nozzles, or guide vanes, that direct flow to the impeller blades. The analysis portion of this investigation took less than a day, compared with multiple days without the use of BladePro-CF and simulation. Impeller failures can cost from $50,000 to $100,000, making the avoidance of these situations of great interest to both the manufacturer and end users.

The combination of BladePro-CF and FEA products from ANSYS allows Praxair engineers to easily and accurately determine the quality of their turbomachine impeller designs by providing an appropriate margin against fatigue damage. Using this approach, they can effectively and quickly predict and examine in detail the vibration-related parameters that could affect the reliability and life of their designs.
Meeting predicted life estimates and avoiding component failure are essential for hydropower generation projects. The New York Power Authority is involved in the rehabilitation of large hydropower units, such as those at the St. Lawrence-FDR Power Project. Recent progress in numerical simulation software allows these units to reach a high level of performance in order to improve productivity and reliability. For phase 2 of the rehabilitation project, eight new replacement propeller runners are to be supplied by Alstom Hydro — a company that develops power generation products and systems — for the original Allis-Chalmers turbines.

These runners are 6.096 meters in diameter, have an expected lifespan of 70 years and are capable of producing 64.9 MW under 24.7 meters net head. The hub and blades of the runner are castings of stainless steel machined to final shape. Dynamic loads that occur during the life of hydraulic turbine runners can cause failure and, therefore, present a significant risk with regard to the reliability of the turbine as well as the economic viability of the project. Because of this, technical specifications now require dynamic load analyses of turbine runners. A fracture mechanics analysis is used to evaluate crack growth rates for flaws and to predict the mechanical failure of the component over a given lifespan. To help ensure longevity, engineers at Alstom Hydro of Quebec, Canada, use ANSYS Mechanical software to compute the stress intensity factor as a function of the crack length for complex shapes like those found in propeller-type hydralic turbine runners.

The goal of fracture mechanics analysis is to determine the critical dimensions of an initial defect at a given location for the expected lifetime of the component. Engineers evaluate crack propagation and brittle failure of the runner by computing stress intensity factors and then applying a loading pattern representative of the anticipated operating conditions. The fracture mechanics approach is, therefore, particularly suitable for the analysis of the partial-penetration welds between runner hubs and blades.
A standard approach to evaluating the stress intensity factors of a defect is to use the British Standard BS 7910:2005: Guidance to methods for assessing the acceptability of flaws in metallic structures. This standard considers the blade–hub junction as a simplified geometry, such as a cruciform joint. In the case of the blade design used in the St. Lawrence runner, as well as some other modern runner designs, however, the shape of the welded joint is far from being cruciform. These partial-penetration welded joints include an unfused area, such that the weld metal does not penetrate the entire cross section of the joint. Engineers at Alstom developed an improved methodology to design a safe weld configuration for these new runners. In parallel, the engineers validated stress intensity factors by comparing results from ANSYS Mechanical software to published solutions for a standard cruciform joint from the British Standard. The results were satisfactory.

Next, the engineers developed a relationship to describe stress intensity factor as a function of the crack length for each plane. They computed the mixed-mode stress intensity factors using a displacement extrapolation method. They evaluated stress intensity factors for a wide range of crack lengths and positions. They computed the mixed-mode stress intensity factors using a displacement extrapolation method. They evaluated stress intensity factors for a wide range of crack lengths and positions. Next, the engineers developed a relationship to describe stress intensity factor as a function of the crack length for each plane. They were then able to use this information to design runners with partial welded joints that would be better suited to withstand dynamic loads occurring over the lifetime of the runner.

This resulted in a reduced volume of welding, thus leading to less welding distortion and a very accurate final geometry. Because of Alstom’s use of FEA in the design process, the efficiency of the runner can be maintained at a high level, cavitation erosion can be avoided and overall global behavior can be improved. The first runner is now operating smoothly, and it is estimated that it will produce green energy for at least the next 70 years. The remaining units will be delivered and commissioned in years to come.

References
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 remeshing 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.

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

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.
Savings from Submerged Combustion Melting

Simulation helps glass manufacturers understand complex phenomena in next-generation melter technology.

By Bruno A. Purnode, Owens Corning Science & Technology Center, Ohio, U.S.A.

The glass industry annually produces 21 million tons of consumer goods valued at $28 billion. Energy costs for this volume of goods account for approximately 15 percent of production costs. Theoretically, glassmaking requires about 2.2 million BTUs of energy per ton of glass, but usually more than twice that amount is actually used due to various system losses. Given this reality, the industry is constantly seeking and developing new ways to lower capital and energy costs. Owens Corning has been melting glass for more than 65 years yet has not stopped working to find better and more sustainable ways of doing so. Over the last 14 years alone, Owens Corning has reduced its global energy intensity by 40 percent through a variety of glass melting innovations. With the possibility of reducing capital cost expenditures by more than half, submerged combustion melting (SCM) technology offers the potential to meet this goal.

SCM was first commercialized a decade ago in the Ukraine for production of mineral wool. The process takes fuel plus oxidant and fires them directly into the bath of the material being melted. The combustion gases bubble through the bath, creating an intense transfer of heat between the two phases. Meanwhile, forced convection-driven shear effects provide rapid particle dissolution and enhance temperature uniformity in the bath. Batch handling systems can be simple and inexpensive because the melter is tolerant of a wide range of raw material sizes; the systems can also accept multiple feeds and do not require perfect feed blending.

Presently, very little is known about the physics of the submerged combustion process because experimental or field data is not easily accessible in such a melting system.
Therefore, in order to support submerged melter designs and to better understand the complicated melting phenomena, an important modeling effort using FLUENT computational fluid dynamics (CFD) software was initiated at the Gas Technology Institute (GTI) in Illinois, U.S.A. The simulation effort, which included model development support from ANSYS, Inc., was part of a larger U.S. Department of Energy–sponsored project being conducted in partnership with a consortium of glass companies, including Owens Corning. The project’s goals were to design, demonstrate and validate the melting stage of a next-generation melting system.

The presence of extremely complex physics and chemistry, as well as the widely disparate time scales between the combustion gases and the glass flows, made solving the full problem impractical. Therefore, the simulation team from GTI, the consortium companies, Owens Corning and ANSYS established a pragmatic, three-stage modeling strategy in order to find a compromise between faithfully describing the process physics and maintaining a reasonable computational cost.

The first CFD modeling stage was a 2-D axisymmetric analysis, which solved the full transient, two-phase gas–liquid submerged combustion problem for a single-burner region. This analysis used the volume of fluid (VOF) multiphase method in FLUENT software to track the gas phase bubbling through the liquid phase; it also used the eddy dissipation model to simulate the combustion. With the eddy dissipation model, the reaction rates were assumed to be controlled by the turbulent mixing, allowing expensive kinetic calculations to be avoided. The team simulated the turbulence itself with the realizable k-ε model and modeled radiation using the discrete ordinates (DO) method, since this is one of the most versatile of all the radiation models and also has a reasonable computational cost.

In the second modeling stage, the CFD group focused on the overall flow and heat transfer in the melter. They extracted equivalent momentum and heat source terms derived from time-averaged VOF results from the first modeling stage and used them to generate a set of user-defined functions (UDFs) to represent the momentum and heat sources in the second stage. The group then used the UDFs to model a subsequent 3-D steady-state, single-phase analysis of the entire multi-burner melter.

The third modeling stage consisted of a 3-D transient tracer species analysis. In this final analysis, the CFD team used the velocity and temperature fields from the second modeling stage to analyze the transport and dissolution of the batch by calculating the residence time distribution.

Preliminary trials in the one-ton-per-hour pilot-scale melter seemed to validate the staged modeling approach. The research team successfully compared temperature, velocity, pressure and residence time simulation results to measurements in fully instrumented submerged melter trials conducted by GTI. A complete validation of the model will be done in the coming months as more trial results become available.

Mathematical modeling of the submerged combustion process using the FLUENT product has been an integral part of the SCM project. Simulation has led to a better understanding of the complex glass and gas flows and thus has been used extensively for designing the melters as well as for guiding their operation. This constitutes a first step toward the industry’s goal of reducing cost and energy usage with a next-generation melting system.
Breathing Easily

Simulation of airflow in human noses can become a useful rhinosurgery planning tool.

By Alexander Steinmann and Peter Bartsch, CFX Berlin Software GmbH[1], Berlin, Germany
Stefan Zachow, Zuse Institute Berlin (ZIB), Medical Planning[2], Berlin, Germany
Thomas Hildebrandt, Asklepios Clinic Birkenwerder[3], Birkenwerder, Germany

A requirement for normal breathing through the nose is an undisturbed passage through the nasal airways. If this condition is not fulfilled due to any obstruction or deformation, surgical correction of the nasal airways might be required. Rhinosurgery is a reconstructive surgical approach that reshapes the nose and/or nasal structure and often is used to correct birth defects or other breathing problems.

To understand the effects of nasal anatomy on normal breathing, a team in Germany composed of members from the Zuse-Institute Berlin, Asklepios Clinic Birkenwerder and CFX Berlin Software GmbH carried out simulations using ANSYS CFX computational fluid dynamics (CFD) software. The research team based the analysis models on highly detailed internal and external nasal anatomy. The ability to simulate complex airflow characteristics with regard to individual anatomy enables the study of the physiology and pathophysiology of nasal breathing on a per patient basis. As a result, fluid flow simulations can become an extremely useful tool in treatment planning for functional rhinosurgery.

For this study, the research team based their investigations on a reference model of the nasal airways created from actual human anatomy without obvious pathologic symptoms. To develop a geometric model for this case, researchers first acquired a helical computed tomography (CT) scan of a male volunteer following local administration of a decongestant. High-resolution tomography with an almost isotropic spatial resolution of $0.37 \times 0.37 \times 0.4$ millimeters allowed for the representation of internal anatomical structures with sufficient detail. This provided the team with three-dimensional geometric information that they used to create a simulation model of the nasal and paranasal cavities.

The research team then used AMIRA®[4] software to reconstruct and
mesh the flow domains based on the CT scan information. In addition to creating the volumetric grid of inner airway structures, the research team reconstructed the facial soft tissue. To accomplish this, they generated a grid for the anterior inflow region in order to simulate the effect of the nose and face external surface geometry on the inhalation flow behavior. Finally, the team exported the meshed model, with locally refined resolution and suitable element quality, in computational fluid dynamics (CFD) General Notation System (CGNS) format for import into ANSYS CFX software.

The CFD simulation involved calculating the transient flow behavior over seven breathing cycles. The researchers applied a pressure difference between the inlet and outlet (lung) as a boundary condition. The lung pressure was represented as a function of time and was derived from a series of active anterior rhinomanometry (AAR) measurements, recorded from the same subject from whom the geometry model was derived in a comparable mucous membrane swelling condition. Samples were gathered at a rate of 2,000 samples per 15-second measurement. Finally, researchers measured air volume flow for validation of the simulation results and found that there was appropriate agreement between the experimental and numerical data.

To gain a better understanding of the relationship between nose morphology and respiration, the research team is currently investigating the effect of anatomical changes to the external nose geometry. Using an advanced biomechanical tissue model, they can vary the shape of the external nose in a realistic manner, increasing or decreasing the nasolabial angle or the cross section of the nasal valve, for example. Such variations may disturb the inspiratory inflow due to an increased resistance, an impaired airflow distribution or a pathological turbulence behavior. In this way, interactive geometry alteration of the nasal airways in combination with a simulation analysis of the resulting fluid flow using ANSYS CFX technology can provide a basis for a sophisticated virtual rhinosurgery planning tool. Conclusions drawn from simulation could have an important impact on future surgical and conservative therapeutic concepts, thus driving clinical research.

In further investigations, the research team will study the humidification of nasal airflow as a multi-fluid flow with an additional transport equation for water vapor. In those studies, the humidity charge of the nasal mucosa will be modeled, and humidity transfer between mucosa and air will be considered by appropriate boundary sources at the fluid walls. To accomplish this, researchers will introduce dense layers of pentahedral prism elements at the air–mucosa interface to ensure accurate numerical calculations for fluid shear stresses, such as changes of air velocity at the mucosal walls. In addition, this simulation methodology will enable the study of pharmacokinetic issues, such as the effective application of drug delivery via the respiratory system. In these simulations, the medication particles and inspired air will be regarded as a multi-phase flow consisting of liquid droplets (dispersed distributed particle flow) in a continuous air stream; a heat transfer mechanism will also be included.

References
Ins and Outs of Inhalers

Simulation helps optimize the performance of a dry powder inhaler for drug delivery.

By A.H. de Boer and P. Hagedoorn, Department of Pharmaceutical Technology and Biopharmacy, University of Groningen, The Netherlands
R. Woolhouse and J. Tibbatts, ANSYS, Inc.

In recent years, there has been growing interest in dry powder inhalers (DPIs) as a drug delivery system that could significantly impact the treatment of diseases. The clinical applications for DPIs now extend well beyond the treatment of lung diseases, such as asthma, chronic obstructive pulmonary disease (COPD) — which includes chronic bronchitis and emphysema — and cystic fibrosis. Recently, extensive media coverage has been given to the introduction of inhaled diabetic insulin, and research is currently being carried out to develop methods for the delivery of antibiotics and vaccines as dry powders. A U.S. Federal Drug Administration report indicated that, unlike other drug products, the dosing, performance and clinical efficiency of DPIs may be directly dependent on the design of the device.

In order to develop biopharmaceutical drugs, like peptides and proteins, for delivery as a dry powder, two significant hurdles need to be overcome. First, the drug must be stabilized in a dry state as particles with an aerodynamic diameter in the range 1 to 5 microns. Second, a dry powder inhaler must be developed that will efficiently deliver these microscopic drug particles to the patient. As a result of research in this area, a team at the University of Groningen in the Netherlands has developed a disposable DPI, The Twincer™, for the delivery of high drug doses. Tests have shown that the Twincer is capable of effectively delivering in a single inhalation a 60-milligram dose of pure micronised colistin sulphotemate, a drug used for the treatment of cystic fibrosis.

The Twincer holds the drug in a blister in which the tiny particles have likely joined together to form cohesive agglomerates. In order to break up these particle agglomerates, which are too large to be effective, the Twincer employs two parallel classifiers, circular chambers that apply inertial and shear forces to the particles through a carefully controlled airflow.

To accomplish this, each classifier has three tangential ports that generate
a swirling flow within the classifier: One port delivers air from the classifier inflow; a second delivers air and entrained drug particles from the drug inflow; and a third port delivers air from the bypass channel, which functions to reduce the pressure loss of the device and, therefore, to control the effort required from the patient to make the desired inhalation. After the air and drug particles mix in the classifier chamber, they de-agglomerate and then exit the classifiers through a small opening in the base, where they join the bypass airflow and continue into the mouthpiece. The performance of the inhaler is assessed in four ways: the consistency of delivered dose, the delivered fine-particle fraction (FPF) within the dose, the particle retention within the device and the pressure drop characteristics of the device.

During experiments using the Twincer, researchers have demonstrated that minute changes in geometric details can have a significant influence on the inhaler’s performance. To further understand the behavior of the inhaler and the reasons for the design sensitivity, the team conducted a number of computational fluid dynamics (CFD) simulations using FLUENT software.

The research team took results from the CFD simulations for pressure loss and compared them with experimental measurements in order to validate the model. The results of both the CFD analysis and experimental work showed good agreement. Concerning the particle trajectories, the CFD analysis predicted that particles 10 microns in size and larger would be retained within the classifiers, whereas smaller particles would exit the classifiers and flow into the mouthpiece. The retained particles would include larger sweeper particles, those typically 100 to 200 microns in diameter. These predicted particle trajectories also compared favorably with experimental observation made using a laser diffraction technique. Finally, the team concluded that the drug particle cut-point is typically between 5 and 7 microns, depending on the properties of the drug and the flow rate.

The maximum dose that can be delivered by the Twincer depends on the flow split — the percentage of the total flow rate that passes through the drug inflow — and the patient’s lung capacity, which may potentially be impaired. Since the design goal of the inhaler is to deliver high drug doses, the flow split within the device is of key importance. The CFD simulations showed that only 16 percent of the total airflow passed through the drug inflow and that 60 percent of the airflow bypassed the classifiers completely. These flow splits remained roughly constant over the expected operating range of the device.

To try to improve the flow split, researchers carried out an additional CFD simulation in which a blockage was introduced into the bypass channel initially using a porous media. The team discovered that by partially blocking the bypass channel, the total airflow was reduced without affecting the mass flow available to mobilize the drug and to generate swirl within the two classifiers. From this observation, the researchers inferred that such a design change is unlikely to affect the

Sweeper particle retention within classifier chambers (approximately 175 micron particles)

CFD simulation showing particle trajectories for 1-micron (top) and 10-micron (bottom) particles
rate of entrainment of the drug from the blister or the rate of particle agglomerate breakup in the classifier. However, such a design change may reduce the particle exit velocity from the inhaler’s mouthpiece, a condition that increases delivery of drug particles to the deep lung.

Using CFD analyses, the research team also learned that the port from the bypass channel to the classifier did not carry any flow, or was a “dead flow channel.” Previously unexplained experimental observations had shown that an accumulation of particles often occurred in these flow channels. The team carried out an additional CFD simulation with these flow channels removed. As expected, the simulation showed that the removal of these flow channels had no significant impact on the performance of the device.

The use of CFD in the development of the Twincer allowed detailed assessment of the flow behavior within the inhaler. The results from the CFD analyses showed good agreement with experimental results and observations. Researchers have used the information gained from the CFD simulations to guide design modifications of both the classifier inlet channels and the bypass flow channel. Following redesign of the inhaler prototypes, researchers again plan to use CFD simulation alongside an experimental program to further optimize the Twincer DPI.
Supporting the Oil and Gas Industry

Longevity and safety of drilling derricks and substructures are increased through stress analysis.

By Luis M. Peñalver, General Manager, Consultora de Ingeniería Peñalver, C.A. (CIPCA), Puerto Ordaz, Venezuela

The oil and gas industry faces challenges associated with age-related equipment deterioration. In these scenarios, companies must ask themselves what to do with the equipment; more specifically, should they maintain the equipment as it ages, or retire it and purchase new equipment? In some cases, the most cost-effective solution involves repairing the equipment in order to extend its useful lifespan. One specific challenge is the repair and recertification of oil derricks used in the drilling process.

There are several types of derricks. The structure of each is suitable for the type of activity, which could be drilling, reconditioning or well cleaning. The most common type is a rigid design that has four legs secured to the corners of a metallic substructure.

Oil derrick design specifications require that the structure is able to support the load of vertical tubes used for the drilling operation; the derrick also must resist wind loads that may have velocities up to 160 kilometers per hour. In addition, during the extraction of the drill pipe, the pipe could get stuck in the well due to irregularities or obstructions in the hole. In these cases, the derrick must resist, within reasonable limits, the force required to release the drill pipe from the hole.

After years of service and perhaps poor operating practices, derricks and substructures involved in drilling operations may begin to exhibit damage such as deformation, fatigue, breakage or misalignment of their structural elements. These problems can be corrected through structural repairs, but the equipment then needs to be recertified for use.

New equipment can be repaired and certified by the original manufacturer. However, much of the equipment currently used in the oil industry has been in operation for more than 10 years. The original derrick manufacturer may no longer exist, therefore, these derricks are considered unidentified equipment. Evaluation made by manual calculation is a long, slow process that does not reach the accuracy level required by industry standards for recertification. In addition, physical variables such as fatigue and remaining life calculations for these large structures are extremely complicated.
OIL AND GAS

According to the American Petroleum Institute standard API-4G (Recommended Practice for Maintenance and Use of Drilling and Well Servicing Structures), load rating for a well servicing or drilling structure of unknown manufacture may be determined by a process including inspection and engineering practices such as simulation. This process may include structural analysis in accordance with API-4F (Specification for Drilling and Well Servicing Structures), which states that the accuracy of standard design ratings of each structure shall be tested by proof loading or by a computer model, such as finite element analysis (FEA), to verify the structure for the design loads.

In order to fulfill these international specifications and to maintain the reliability and structural integrity of equipment during operation, Consultora de Ingeniería Peñalver, C.A. (CIPCA) in Venezuela uses FEA software from ANSYS to examine various clients’ drilling structures. CIPCA employs simulation to determine maximum load capacity, stress distribution on the structure, regions prone to failure (critical regions), potential life cycle, fatigue effects and load rating. The ability to determine stress in a structure with more than 20 years’ service and for which the manufacturer no longer exists gives oil and gas companies the ability to comply with international specifications and to rescue equipment that would otherwise need to be retired.

CIPCA applied this approach to equipment owned by COMANPA, C.A., an oil and gas drilling company also located in Venezuela. The derrick for the oil rig named COMANPA 27 was built in 1971 and was designed with a maximum lift capacity of 180,000 pounds. After years of service, the derrick showed general deformation in its primary and secondary structural elements and was tagged unusable for drilling service after failing inspection. In order to evaluate the derrick under the API specifications, CIPCA began by modeling the geometry of the existing derrick, including deformed parts, with ANSYS DesignModeler software. Shell elements were used to model the major structural members. This saved computational time in comparison to using solid elements to mesh these relatively thin parts. In addition, better accuracy could be obtained for the same number of degrees of freedom by utilizing shell theory for thin geometries. Also, joints were modeled with edge-to-edge contact elements that were automatically detected between the structural members through ANSYS Workbench simulation.

By using ANSYS contact technology to deal with interactions between parts, mesh densities could be adjusted appropriately without the mesh of one part influencing the mesh density of the adjacent part. The FEA simulation, which included fatigue effects, was performed using ANSYS Mechanical software within the ANSYS Workbench environment.

From the results, CIPCA concluded that the derrick could not operate at its original design specifications; however, it was determined that the derrick could operate safely with a modified lift capacity of 100,000 pounds. Following analysis, the derrick was able to complete scheduled drilling for an additional two years.

Having a drill out of service can significantly impact costs and, therefore, the bottom line, according to AKERE ENERGY, C.A., another company that specializes in oil and gas exploration, drilling and operation based in Venezuela. It is very important for such companies to meet their annual drilling schedule, as these companies maintain contracts to drill a specified number of holes in a region. Derricks that are out of service for certification can only be replaced by equipment that has been previously contracted and certified. If equipment is not available, then production goals are not met, resulting in loss of profit.
At Modine Manufacturing Company in Wisconsin, U.S.A., designing charge air coolers and exhaust gas recirculation (EGR) coolers is largely driven by emerging emission standards in the United States and Europe. These new emission standards equate to higher thermal loads and more rigorous durability requirements for automotive engine and exhaust components. With the surge in demand for engine components that meet the new emission standards, it was apparent to Modine that it needed to conduct a greater number of concurrent engineering design projects without significantly impacting engineering headcount or physical test facility capital investment. Physical testing is expensive, and test capacity limitations make it nearly impossible to turn around a design project quickly and efficiently through testing alone. Given these factors, the need for leveraging the virtual environment was clear.

In order to deliver a quality product that would fully satisfy its customers, Modine deemed they would need to restructure their development process while also pursuing customized analysis tools tailored to this process. Ideally, these tools would be readily usable by product design staff as well as the traditional analysis staff. Equipping design engineers with straightforward, customized analysis tools would allow more virtual analyses to be executed without the need to increase the number of analysis engineers.

To help modify the process, Modine’s management decided to co-locate all analysis engineers in one physical location in the middle of each of the product line groups, forming the Virtual Technology Group. By co-locating analysts from all product lines, this new team could share technology processes and create best practices for rolling out simulation technologies to engineers in different divisions. Modine selected tools to equip the Virtual

Integrated Analysis Achieves State-of-the-Art Workflow

A collaborative process and better tools help Modine engineers leverage the virtual environment to meet emission standard design changes.

By Allan Wang, Manager Virtual Technology Group, Modine Manufacturing Company, Wisconsin, U.S.A.
Shane Moeykens, Strategic Partnerships Manager, ANSYS, Inc.
Technology Group — including Pro/ENGINEER® 3-D computer-aided design (CAD) from Parametric Technology Corporation (PTC) and FloWizard software from ANSYS for analyzing fluid dynamics; these tools have been integrated by ANSYS to allow for seamless geometry transfer.

Disseminating analysis tools in a usable form throughout an organization is one approach for increasing productivity. However, traditional analysis tools can come with a substantial learning curve. In contrast, the integrated Pro/ENGINEER FloWizard configuration offers a significant level of automation and was appropriately customized to specific design activities at Modine, enabling these tools to be readily used by product designers. For example, a design engineer working with liquid-cooled charge air coolers could rely upon software customization unique to this application for greater efficiency and ease of use.

Describing Modine’s new process, Dr. Jonathan Wattelet, global director of research, said, “Everything is going toward simulation to improve product development processes. We expect the number of engineers we employ to remain relatively constant. We need to use tools such as Pro/ENGINEER and FloWizard software to make our engineers more efficient. By 2010, we’d like one-half of our engineers to be using tools like the integrated Pro/ENGINEER FloWizard solution.”

The process changes Modine implemented had an immediate impact on the design of their products. In one example, Dan Raduenz, an application engineer from the Engine Product Group, used Pro/ENGINEER and the FloWizard product to optimize the flow and pressure drop in a liquid-cooled charge air cooler. Raduenz set up five Pro/ENGINEER versions of his design and in 20 minutes had created his CFD cases using FloWizard. He then processed his CFD solutions overnight and, by the next morning, was able to determine the best path for design and manufacturing. Raduenz estimated that this design would have cost a minimum of $10,000 for just two iterations using traditional methods and tools, not to mention the significant additional queue time and setup time needed to perform physical testing in lieu of these virtual simulations.

Dave Janke, supervisor of the Virtual Technology Group, estimated that each preliminary internal flow analysis of the liquid-cooled charge air cooler using the current processes costs Modine $400 and approximately one day of time, compared with several days using the conventional high-end CFD tools. The end result of the organizational and process change is that the new designs consistently meet and/or exceed Modine customers’ requirements for performance, quality, cost and delivery. “This is a clear case of cutting-edge simulation. FloWizard software’s automation and its interoperability with Pro/ENGINEER were critical for this labor reduction,” stated Scott Wollenberg, managing director of the Global Engine Product group.

Modine’s senior management has made the commitment to provide their organization with the right environment and tools to enhance communication and productivity. Dr. Anthony C. De Vuono, vice president and chief technology officer, commented, “I am thrilled with the way the organization has embraced the environment. Managing directors of Modine’s product line groups are positive and are reporting exceptional productivity all around. Computer simulation and analysis allows us to use our physical test facilities more wisely, facilitating these productivity gains.”
From CAD to CAE

FLUENT software now offers support for Autodesk Inventor.

By Shane Moeykens, Strategic Partnerships Manager, ANSYS, Inc.

A seamless flow from computer-aided design (CAD) to computer-aided engineering (CAE) has long been an important goal for engineering design teams seeking to increase productivity and reduce costs. Now, companies who use Autodesk® Inventor™ have streamlined access to selected CAE software from ANSYS. The new Fluent Inventor Connection provides users with the ability to automatically launch FloWizard or GAMBIT software from the Inventor environment. With a single click, users can load a geometry model directly into FloWizard or GAMBIT to set up a computational fluid dynamics (CFD) analysis and solve for the flow in or around the geometry. The Fluent Inventor Connection provides seamless connectivity between design and analysis processes.

As a consultant and longtime user of ANSYS software, Brad Stevens of Kx Simulation Technologies sees benefits customers can reap from the tight integration of design-level analysis tools. “We are certain that the addition of even tighter integration between the ANSYS design-level CFD tool, FloWizard, to Autodesk Inventor will bring increased productivity and value to Inventor users,” Stevens said. Kx, a Cincinnati, Ohio-based engineering consulting firm, delivers customized solutions backed by more than 15 years of engineering experience.

Prototyping can account for as much as 25 percent of the entire development cycle. “Digital prototyping offers an efficient and cost-effective alternative to fabricating costly prototypes. Virtual testing and analysis allow a company to validate basic design properties in the early phases of the drafting and development process, providing further development cycle and cost benefits,” said Robert “Buzz” Kross, vice president of Autodesk Manufacturing Solutions. Autodesk has had a formal relationship with ANSYS, Inc. since 2005, through the joint solution for stress and strain analysis powered by ANSYS DesignSpace technology. “We’re very excited to expand this relationship to other areas of multiphysics, such as the CFD capability enabled by the Fluent Inventor Connection and FloWizard 3.0,” Kross added.

The Fluent Connection software also provides interfaces with UGS NX™, Pro/ENGINEER® Wildfire® and SolidWorks®. Integrating core CAE technologies with independent design tools has been a key part of the ANSYS strategy for nearly a decade.
Analyzing Buckling in ANSYS Workbench Simulation

Simulation shows how parts catastrophically deform under compressive loads that exceed the structure’s material strength.

By Sheldon Imaoka, Technical Support Engineer, ANSYS, Inc.

One problem faced in the design of structures is buckling, in which structural members collapse under compressive loads greater than the material can withstand. Examples include the local failure of a box girder of a bridge or an aluminum beverage can crinkling when compressed. Figure 1 shows a plastic bottle deforming in this manner under an internal pressure.

Using ANSYS Workbench Simulation functionality provides many tools to aid users in solving geometric instability problems, ranging from linear (eigenvalue) buckling to nonlinear, post-buckling analyses. Eigenvalue buckling analysis is a good approximation technique that, although less precise than nonlinear buckling analysis, is a relatively quick and easy way to determine, for example, critical loads that induce buckling and possible buckling modes (that is, the different ways the structural member can deform). The solution time for eigenvalue buckling typically is significantly faster than a nonlinear buckling analysis, meaning that a great amount of useful information comes at a relatively cheap computational price.

Performing Basic Linear Buckling Analysis

ANSYS Workbench Simulation allows users to easily set up linear buckling analyses. First, a user must set up the loads and boundary conditions under a Static Structural...
analysis branch. Then the user must add a second analysis branch, Linear Buckling. In this step, the Initial Conditions branch references the Static Structural branch, so that loads, boundary conditions and the stress state of the system can be obtained.

Under the Analysis Settings branch, the user can request any number of buckling modes. While the default is to solve the first buckling mode, the author recommends solving for three or more buckling modes in order to verify whether or not there may be multiple buckling modes that could be triggered.

After solution, the buckling mode shapes and load multipliers can be reviewed. The magnitude of all of the loads defined in the Static Structural branch multiplied times the load multiplier provides an estimate of the critical load.

Including Initial Imperfections

If a user considers symmetric geometry, even a non-linear buckling analysis may predict too high a critical load. Consider a simple plate simply supported at one end (A) and guided on the other (B) with a compressive load (C), as shown in Figure 2. Although a user may assume that buckling should occur in the out-of-plane direction, this may not occur if the geometry is modeled perfectly.

To correct for this, use a buckled mode shape calculated from a linear buckling analysis to create a small imperfection or perturbation in the mesh for use in nonlinear buckling analyses. This can be accomplished in ANSYS Workbench.

Simulation using a Commands object with the UPGEOM ANSYS command.

All result files are contained in the Simulation Files folder under subdirectories, such as Linear Buckling. To use a buckled mode shape to perturb the geometry, first determine the buckled mode shape as well as the maximum amplitude. In the nonlinear static analysis branch, insert a Commands object with the following Advanced Parametric Design Language (APDL) commands:

!/PREP7
UPGEOM,factor,1,mode,’..\Linear Buckling\file’,rst
/SOLU

Note that in this command, factor will be multiplied to the buckled shape mode and the nodes will be moved to new locations. For example, a user may want to perturb the mesh using the first buckled mode shape, which may have a maximum amplitude of 0.5. Using information such as manufacturing tolerances or a given percentage of the thickness of the part, the user may wish to include an imperfection with a maximum value of 0.002. The user then could use the following commands to include the first buckled mode shape:

!/PREP7
UPGEOM,0.004,1,1,’..\Linear Buckling\file’,rst
/SOLU

Figure 2. Plate in this buckling example is simply supported at one end (A) and guided on the other (B) with a compressive load (C).

Figure 3. Plot of displacements in out-of-plane direction.
When using commands in ANSYS Workbench Simulation, note that the system of units should not be changed. The ANSYS result files will be based on the active units when the Linear Buckling analysis was performed. Also, because the mesh is being modified directly by the ANSYS mechanical solver, ANSYS Workbench Simulation will not display the updated nodal position; this should not pose a significant problem in post-processing.

The aforementioned simply supported plate was loaded in-plane with and without an imperfection, based on the first buckled mode from the eigenvalue buckling analysis. Figure 3 shows the plot of displacements in the out-of-plane direction. Note that without any imperfection, no buckling occurs. With the small imperfection, buckling occurs at approximately 85 percent of the applied load.

Capturing Post-Buckling Behavior

In situations such as failure analysis, post-buckling behavior must be studied. Techniques such as solving the system as a transient analysis or using the arc-length method have been available in mechanical simulation solutions from ANSYS for a very long time. A relatively new method introduced in ANSYS 11.0 technology is the nonlinear stabilization technique. This method is controlled with the STABILIZE command and is easy to implement through ANSYS Workbench Simulation.

Conceptually, nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Before the critical load is reached, the system typically may have low displacements over a given time step. This can be thought of as a low pseudo velocity that would not generate much resistive force from the artificial dampers. On the other hand, when buckling occurs, larger displacements occur over a small time step; as a result, the pseudo velocity becomes large and the artificial dampers generate a large resistive force.

Nonlinear stabilization can be specified either by entering a damping factor or energy dissipation ratio. The ratio typically ranges from zero to 1 and can be thought of as the ratio of work done by the damping forces to the potential energy. When this method is used, the effective damping factor is printed for reference purposes in the Solution Information solver output as follows:

```plaintext
*** DAMPING FACTOR FOR NONLINEAR STABILIZATION = 0.1840E-01
```

Because of the easier interpretation of the energy dissipation ratio value, it is recommended that users first select values closer to 0, reflecting less damping. Other controls with nonlinear stabilization include using constant values, or ramping the stabilization forces to zero at the end of the load step, as well as selecting at which point nonlinear stabilization is activated.

To use nonlinear stabilization, a user simply needs to insert a Commands object under the Static Structural branch with the STABILIZE command and relevant arguments. For example, to use 0.01 percent constant energy dissipation ratio, one can use the following command:

```plaintext
STABILIZE,CONSTANT,ENERGY,1e-4
```

Note that because nonlinear stabilization can be turned off or used only for certain load steps, a user may wish to separate the load history in multiple steps via the Analysis Settings branch; following that step, the user then activates nonlinear stabilization only when needed through the Details view of the Commands object. Figure 4 shows a tubular system loaded in compression in which post-buckling behavior is captured using nonlinear stabilization.

Contact the author at sheldon.imaoaka@ansys.com for the complete paper from which this column is excerpted.
Parametric Design Analysis for Evaluating a Range of Variables

Tools help to study engineering trade-offs in Simulation Driven Product Development.

By Pierre Thieffry, Product Manager, ANSYS, Inc.

A major challenge in product development is balancing competing engineering requirements. Components often must be lightweight yet strong enough for maximum durability, for example. Users are thus faced with the tedious and time-consuming task of running multiple simulations to find a solution that satisfies most of the requirements.

Fortunately, tools are available to help designers perform parametric analyses in which simulation software automatically solves for entire ranges of specified variables and generates displays that enable users to readily spot trends and identify an optimal design. By clearly showing the relationship of multiple variables and their effect on performance, parametric analysis can guide the product development process to a design configuration that might not have been considered with pure point-solution simulation or that would have proved too time-consuming if individual analyses were manually performed.

Geometric Parameters: A Key to Design Variations

A wide variety of parameters, such as material properties, can be varied to study the impact of those changes on the design, but a major source of variation is the geometry itself. While parametric computer-aided design (CAD) models have existed for decades, very few simulation tools allow them to be used effectively. Some tools, such as the ANSYS Parametric Design Language (ADPL), allow users to create parametric geometries, though the time required to set up such a model increases significantly with the complexity of the geometry.

Typically, one of the most efficient ways of dealing with geometric parameters is provided by the ANSYS Workbench platform, which enables parameters of the CAD model to be driven directly from simulation. The ANSYS interface for major CAD systems not only reads in the geometry data but also imports the geometric parameters, along with attributes or material data in some cases. In this respect, the ANSYS Workbench environment provides an easy solution for defining and
performing parametric analyses. The additional amount of work required to move from a single point simulation to a full parametric analysis is no more than a dozen mouse clicks.

Benefits of Parametric Design Analysis
Parametric analysis is an excellent way to get accurate information about the influence of all parameters on the design objectives, such as system performance with respect to stress, heat flow, mass flow and other variables. With this information, the design team can make informed decisions throughout product development, especially in the early conceptual stage. As a consequence of the parametric analysis, the design team also can react quickly to any modification due to external constraints (for example, manufacturing) and can easily answer any “what if” questions.

Data Representation
Data representation is crucial in order to maximize the benefit of a parametric analysis. Tools such as ANSYS DesignXplorer software are based on response surface methods (RSMs) that help allow users to readily visualize and evaluate performance variations over the entire design space. Such approaches can be applied to any simulated physics applications including structural, computational fluid dynamics and multiphysics analyses. These methods are efficient in terms of computation time, since they use a limited sampling of the parametric space to build the response surfaces, which depict the mathematical relationship between input and output parameters. These 3-D color-coded contours readily convey large amounts of data that would otherwise be overwhelming to decision-makers, who will more easily be able to interpret a simple curve or plot than a list of numbers.

Among other graphical representations, the trade-off plot is probably the first to be considered. It represents the feasibility of a given design: A large sampling (10,000+ points) is performed on a response surface and performances of design variables are plotted. The accompanying trade-off plot indicates, for example, that for the product under analysis to be designed with a safety factor of 1.0, product mass will be in the range of 0.6 and 1.2 kilograms. If the product requirements specify a mass of 0.5 kilograms, therefore, one can see that there are no design solutions that can maintain a safety factor of at least 1.0 at that mass and with that specified design. A new design would have to be generated. On the other hand, if the product requirements specified a mass of 1.0 kilograms, the engineer knows that there is at least one solution. Other tools then exist to help users find the right specific solutions.

Simple 2-D plots (single or multivariate) are the easiest graphs to understand and convey information about the variation of performance with respect to the design variables. Sensitivity charts (bar or pie charts) also provide immediate information about the weight of each of the design variables on the product performance. They enable the engineer to identify the key parameters and know where the focus should be. This type of approach helps to reduce and eliminate time or money wasted on variables that do not influence the design.
Get Inspired!

2008 International ANSYS Conference: Inspiring Engineering

Pittsburgh, PA, U.S.A.
August 26–28, 2008

Join us at the 2008 International ANSYS Conference to discover how simulation can lead to truly inspired engineering.

This exciting event will be the first of a series of ANSYS conferences occurring around the world in 2008.


www.ansys.com