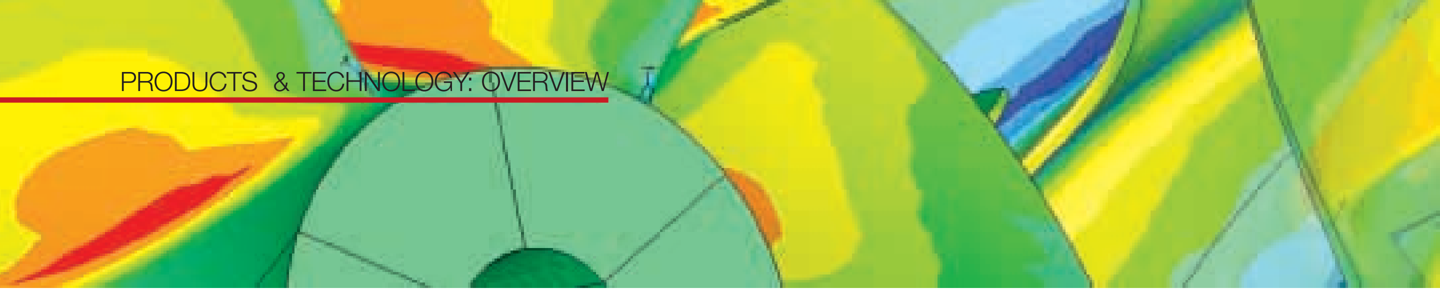
Engineering **Simulation** for the **21st** Century

Five key principles guide the development of simulation products and technology at ANSYS.

By Chris Reid, Vice President, Marketing, ANSYS, Inc.

Technology is the lifeblood of ANSYS, Inc., and the basis for everything we offer our customers. For more than 35 years, ANSYS has been a pioneer in the application of finite element methods to solve the engineering design challenges our customers face. During that time, the evolution of our industry, products and technology has been nothing short of amazing. Fueled by a corresponding increase in the power-to-price ratio of the computing world, the problem size and complexity of simulations have grown to impressive dimensions. The net effect of this is evident in almost every facet of life — from the cars we drive to the energy we use, the products we buy, the air we breathe and even the devices we insert into our bodies to maintain our health.

How have we accomplished this near 40-year run of groundbreaking achievement in engineering simulation and modeling? Staying true to our vision and strategy has certainly been a major factor. Unlike others, ANSYS has never wavered from its core business of engineering simulation software. Instrumental to that vision is our commitment to advanced technology — the cornerstone of our business and



value to our customers. After all, it is our products and technology that enable companies to create the most innovative and globally competitive products for their industry.

Also instrumental to our vision are five principles that guide the development of our products and technologies. The first is *unequalled depth*. Simply stated, for each of the key areas of simulation and modeling technologies — whether it be mechanical, fluid flow, thermal, electromagnetics, meshing or others — we offer a depth of capability that is second to none. This depth has been created over time by reinvesting in the research and development of new technologies, and supplemented by key acquisitions and partnerships along the way. Today, the results speak for themselves in the richness of what we offer our customers, regardless of their specific simulation requirements.

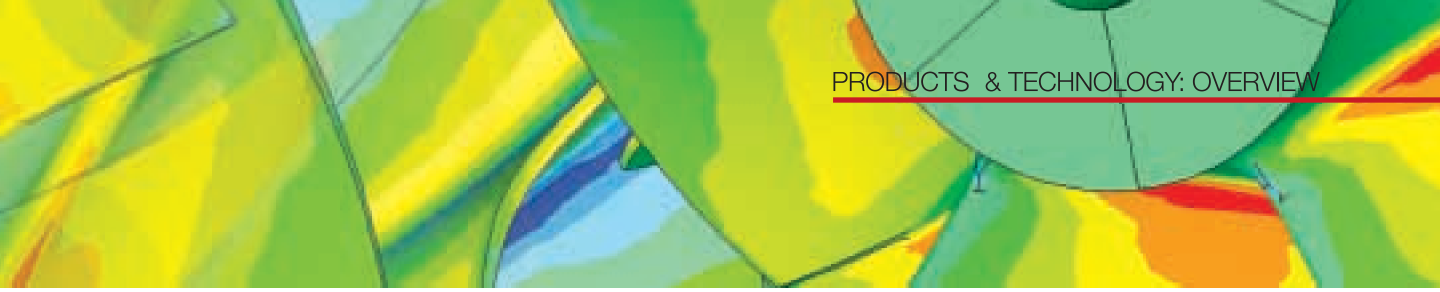
The second guiding principle is *unparalleled breadth*. In this regard, ANSYS has assembled a complete range of simulation capabilities — from preprocessing to multiple physics to knowledge management. Our customers see this as a tremendous benefit, because they know we can provide a solution for each specific area of analysis and that we provide rich depth across our entire portfolio of products and technologies. Some companies, perhaps, can lay claim to this in one or two areas, but we offer this depth and breadth for the full range of simulation and modeling techniques.

In offering both technological depth and breadth, our customers are able to run simulations that are more sophisticated, more complex and more representative of the real world. Utilizing such a *comprehensive multiphysics* approach — our third guiding principle — enables engineers to simulate and analyze complete systems or subsystems using true virtual prototyping. Increasingly, companies realize that a multiphysics approach is essential to attain the most accurate and realistic simulation of a new product or process design. At ANSYS, we not only provide the technologies to do this, but we make them all interoperable within the unified ANSYS Workbench environment. Thus, the user can configure a multiphysics analysis and avoid the need for cumbersome file transfers or intermediate third-party software links. Our technology inherently provides the infrastructure, saving implementation time while providing measurable benefits in speed and robustness as well.

The old adage "one size fits all" is certainly not the case in the world of engineering simulation. Despite the common threads that appear everywhere simulation is used, there are also real differences. Some industries, such as automotive and aerospace, are mature in their use of these tools, while others, such as healthcare, are relative newcomers. Companies within the same industry can be at markedly different stages of adoption, and users within any one company may have vastly different needs or experience with simulation tools. There is a need for flexibility. Customers must be able to adopt the appropriate level of simulation and know they will have latitude in how they move forward.

At ANSYS, we call this *engineered scalability* — guiding principle number four. Why "engineered"? Our scalability is by design and is specifically engineered into the technology we have developed. The depth of our technology allows customers to choose the appropriate level of technology for their needs yet scale upward as their requirements evolve and grow. If the customer is a small company with just a desktop or modest compute resources, or if it is large with hundreds of machines in large-scale compute clusters, our software runs efficiently and brings value. In a similar vein, if the number of users is very small or in the hundreds, scalable deployment has been factored in. Likewise, if the customer is an infrequent user, a designer who wants to perform a simple simulation or an expert analyst, we have the appropriate level of tool for each of those levels of experience. Underpinning this seamless range of capability — from the automated to the most sophisticated and customizable — is the same advanced technology, scaled up or down accordingly.

Technology isn't of much use to the customer if it's extremely inflexible to apply, scalable or otherwise. All of it must be usable in a way that makes sense for the company and its design and development processes, as well as alongside other programs it may have selected for their engineering systems strategy. The vision needs to be flexible and adaptive, not rigid and constraining. In this regard, ANSYS adheres to a fundamental tenet of *adaptive architecture* — the fifth



guiding principle. We recognize the mission-critical nature of what we provide and also how crucial it is that our technology fits within the customer's overall system. There can be CAD systems, selected third-party codes for niche applications, or legacy and in-house software, all of which remain critical components of the overall process. We need to coexist with these and, in fact, enable them to be included in the overall workflow as painlessly as possible.

Many companies are investing in product lifecycle management (PLM) systems. These constitute a major investment and require data exchange with the simulation software. The ANSYS Workbench platform and the new ANSYS Engineering Knowledge Manager (EKM) technology are designed to provide functional coexistence with PLM systems, which actually improves their value to the customer. *Adaptive architecture* means what it says — ANSYS products and technology can adapt to the customer's specific situation. We can be the backbone, coexist peer-to-peer or be a plug-in, whatever the need may be.

Five simple phrases — *unequalled depth, unparalleled breadth, comprehensive multiphysics, engineered scalability and adaptive architecture*. These five tenets are what drive our product development strategy with every dollar we invest. We also think they are the reason that 96 of the top 100 industrial companies on the *FORTUNE* Global 500 list, as well as another 13,000 customers around the world, use technology from ANSYS. The ANSYS simulation community today is the world's largest, and by continuing to pursue our strategy of Simulation Driven Product Development and adhering to these five guiding principles, we see no reason why our vision of placing simulation tools in the hands of every engineer shouldn't become a reality in the near future. ■

Images courtesy Tusas Engine Industries, Silesian University of Technology, Cummins, Inc., FluidDA nv, Modine Manufacturing Company and © 2008 Owens Corning Science & Technology, LLC and the Gas Technology Institute. Used with permission.

Putting **Engineering Knowledge** to Work

New technology enables efficient sharing of rich simulation information and provides enterprise-wide benefits.

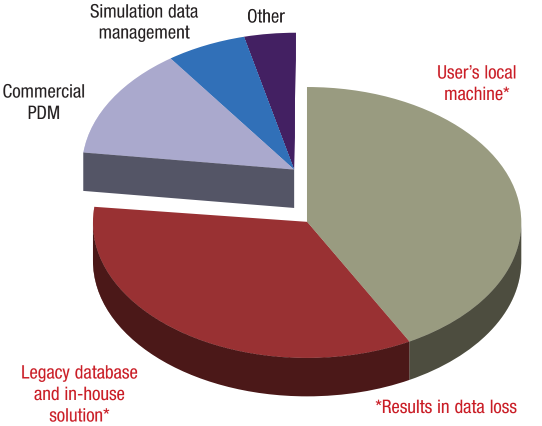
By Michael Engelman, Vice President, Business Development, ANSYS, Inc.

The last three decades have witnessed the evolution of computer-aided engineering (CAE) from a tool used by analysts in a research and development department to one that is integral to the entire product design and lifecycle process. Companies around the world are making greater use of upfront analysis and complex system simulation. With the ongoing integration of CAE into the design process, the focus is shifting from technology issues such as improved simulation techniques, physics modeling and ease-of-use to usage-focused questions such as "How do I better manage and share the voluminous data that is being generated?" or "How do I better capture the engineering expertise that the simulation results represent?" The answer to these questions is often referred to as simulation process and data management (SPDM).

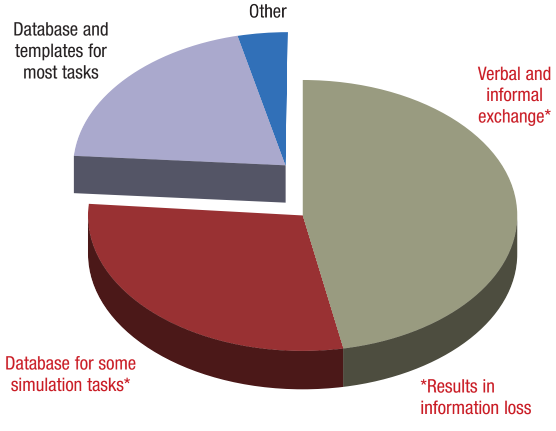
SPDM presents a whole new set of challenges to CAE practitioners. The current focus centers on accessibility — that is, how do the right people get the right data at the right time? More often than not, keeping track of this information is left to the individual analyst or engineer who generated it, so typically at the end of a project it is buried in obscurity somewhere on a hard drive. According to a recent survey by Collaborative Product Development Associates (CPDA), 47 percent of all simulation results are stored on the engineer's local workstation. This valuable intellectual property is usually lost forever when individuals leave a team or company. As anyone who has tried knows, tracking down old data files and analysis models from past simulation projects is difficult or often impossible, even for the people who created them. Consequently, simulations are often redone from scratch — rather than by performing a simple modification to an existing case or model. The result is a significant loss in time and productivity.

With rapid globalization now permeating all aspects of many companies' operations, engineering groups located at different locations around the world constitute virtual 24-hour-a-day development organizations. Effective collaboration and communication are essential to support this global mode of operation. Ineffective communication hurts the entire team, from the engineer who is trying to explain design challenges and concerns to his or her manager, to the teams that need to consider external factors affecting their development process and design. Using tools that can help convey simulation results to all members of a team at every level across the enterprise — regardless of their technical background — can dramatically boost the effectiveness of the team, the product development process and, finally, the quality and performance of the product.

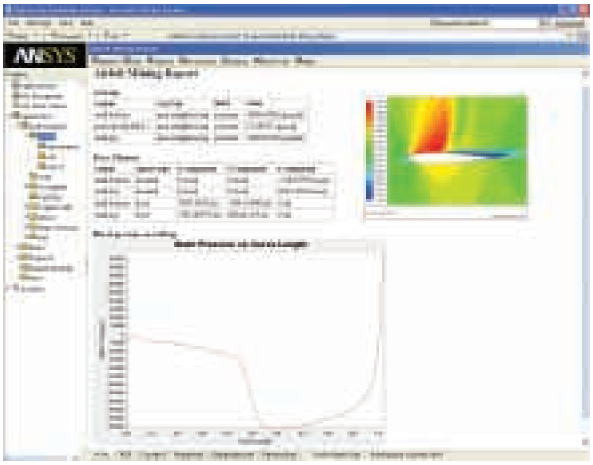
Corporate knowledge is a key business asset in a company's quest for innovation and competitive advantage. Creating, capturing and managing a company's simulation



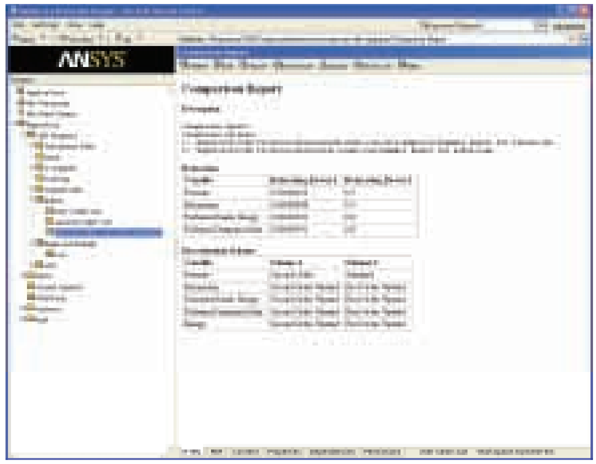
Data management methods



Knowledge exchange methods



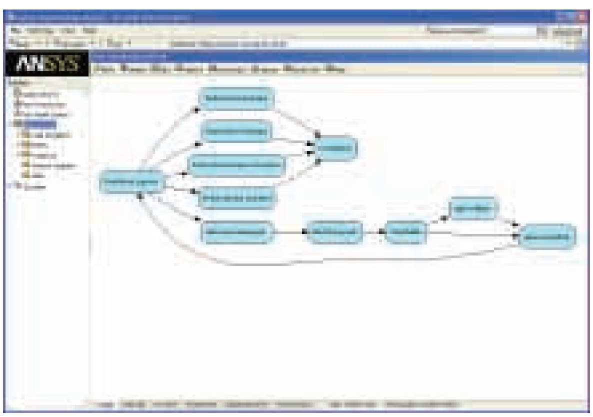
Pressure data for airflow over an aircraft wing is extracted using ANSYS EKM data mining capabilities.



Comparison reports provide users, even those without a technical and simulation background, with the ability to examine simulation results.

expertise is critical to enabling innovation. It empowers users to build on previous experience and fosters continual improvement and collaboration of the expert analysts and design engineers. Effective process management tools that capture simulation best practices, deploy managed simulation tasks and processes, and plug into internal applications within a unified environment are essential to achieve these goals — though they must also require minimal effort and maintenance costs.

Managing simulation data and processes within this context is a specialized subset of the broader product lifecycle management (PLM) vision. This discipline is based on the digital management of all aspects of a product's lifecycle, from concept and design through manufacture, deployment, maintenance and eventual disposal. Unfortunately, those needs that are specific to simulation and SPDM are often overlooked or poorly addressed by today's PLM systems. This is a result of SPDM being more demanding than the file/document-centric approach of PLM and related product data management (PDM) systems. Simulation data is richer, more complex and typically many orders of magnitude larger than other types of product data. An SPDM system is complementary to a PLM system and can add significant value when designed to work in close conjunction with PLM.



Process workflows can be mapped out and displayed in diagram style, as shown here. Among other possibilities, each step may be customized by the user to include automated substeps, assign team members tasks and iterate to the next step.

The ANSYS Engineering Knowledge Manager (EKM) technology, now in its initial release, is aimed at meeting these challenges with extensive capabilities: archiving and management of simulation data, traceability and audit trail, advanced search and retrieval, report generation and simulation comparison, process/workflow automation, and capture and deployment of best practices. It is a Web-based SPDM framework aimed at hosting all simulation data, workflows and tools, whether in-house or commercial, while maintaining a tight connection between them. While providing seamless integration with simulation products from ANSYS — including the ability to automatically extract and organize extensive information about ANSYS software–based simulation files when they are uploaded into the repository — the ANSYS EKM tool is an open system that can manage any type of in-house or third-party simulation products, files or information as well. Moreover, it is a scalable solution that can be effectively used by small workgroups, distributed teams of engineers or the entire enterprise.

With tools and developers that have histories stretching back to the formative years of simulation, ANSYS understands the complexity and challenges of simulation. ANSYS EKM technology was created with an appreciation that access to simulation; developing effective processes for incorporating simulation into individual, workgroup and enterprise-wide efforts; and managing simulation efforts within a larger development or industrial process is a complicated effort — one that can be made simpler. Having access to the right tools, developed by a team that has devoted years to understanding the challenges of simulation, can streamline the incorporation of virtual product development efforts into traditional workflows and environments. Adding ANSYS EKM tools to the capabilities of the family of products from ANSYS empowers organizations of all sizes to better achieve the goal of Simulation Driven Product Development. ■

Applying Six Sigma to Drive Down Product Defects

Probabilistic design and sensitivity analyses help engineers quickly arrive at near-zero product failures in the face of wide manufacturing variabilities and other uncertainties.

By Andreas Vlahinos, President, Advanced Engineering Solutions, Colorado, U.S.A.

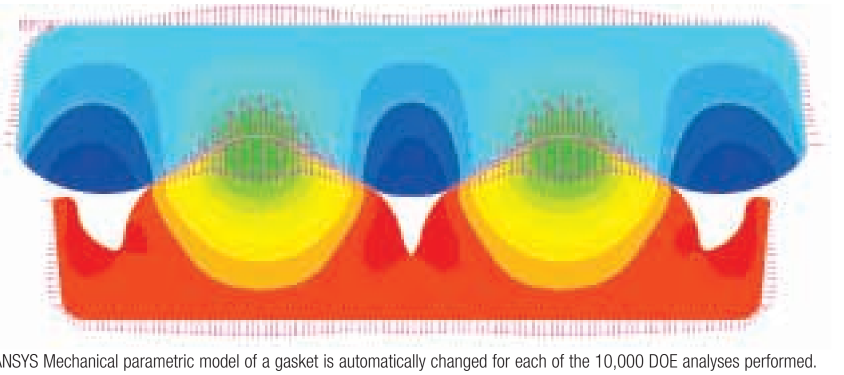
Andreas Vlahinos

Companies often are focused primarily on time-to-market, but the advantages of fast product introductions may be quickly overshadowed by the huge cost of poor quality, resulting in product recalls, rework, warranty payments and lost business from negative brand image.

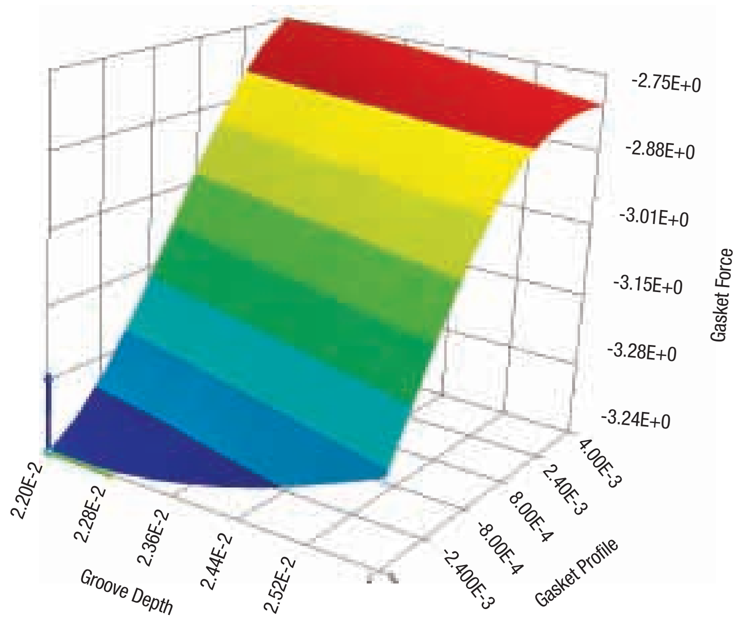
In many cases, such quality problems are the result of variations in factors such as customer usage, manufacturing, suppliers, distribution, delivery, installation or degradation over the life of the product. In general, such variations are not taken into consideration as part of the development of the product. Rather, the integrity and reliability of a design is typically based on an ideal set of assumptions that may be far removed from actual real-world circumstances. The result is a design that may be theoretically sound but riddled with defects once it is manufactured and in use.

Design for Six Sigma (DFSS) is a statistical method for radically reducing these defects by developing designs that deliver a given target performance despite these variations. The approach is a measure of quality represented as the number of standard deviations away from a statistical mean of a target performance value. Operating at three sigma translates into about 67,000 defects per million parts, performance typical of most manufacturers. A rating of six sigma equates to just 3.4 defects per million, or virtually zero defects.

Achieving this level of quality requires a focused effort upfront in development, with design optimization driven by integration of DFSS into the process and rigorous use of simulation. In such DFSS efforts, ANSYS DesignXplorer software is a particularly valuable tool. Working from within the ANSYS Workbench platform and in conjunction with ANSYS Mechanical and other



ANSYS Mechanical parametric model of a gasket is automatically changed for each of the 10,000 DOE analyses performed.



ANSYS DesignXplorer software generated a response surface showing sensitivity of each input variable to contact force.

simulation software, the program performs Design of Experiments (DOE) and develops probabilistic design analyses functions to determine the extent to which variabilities of key parameters impact product performance.

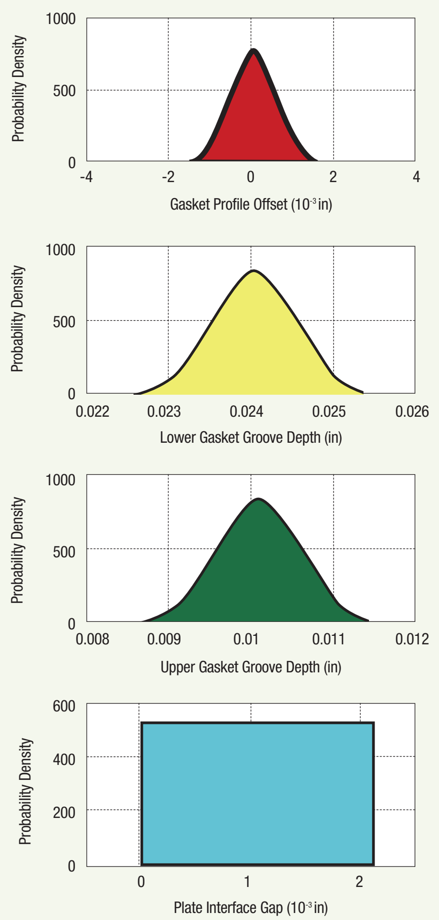
The process is accomplished in four major phases: process automation, design exploration, design optimization and robust design. Utilizing the ANSYS Workbench environment, process automation ensures that simulation tasks are well defined and flow automatically to extract and evaluate key performance variables.

ANSYS DesignXplorer software then performs the DOE, running numerous (usually thousands) analyses using various combinations of these parameters. The ability to quickly and effortlessly execute such an extensive study on this wide range of parameters allows users to perform quick and accurate what-if scenarios to test design ideas. In this way, design exploration — combined with knowledge, best practices and experience — is a powerful decision-making tool in the DFSS process.

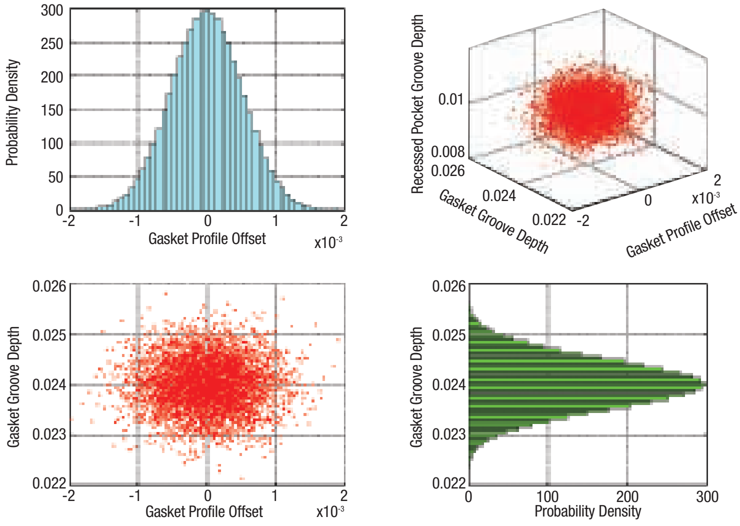
Next, design optimization is performed with the ANSYS DesignXplorer tool in order to select the alternative designs available within the acceptable range of performance variables. Design parameters are set to analyze all possibilities — including those that might push the design past constraints and violate design requirements. Finally, robust design is performed, arriving at the best possible design that accounts for variabilities and satisfactorily meets target performance requirements.

Throughout the process, ANSYS DesignXplorer software employs powerful sampling functions and probabilistic design technology. It also provides valuable output in the form of probability design functions, scatter plots and response surfaces that are critical in DFSS. Seamless interfaces with parametric computer-aided design (CAD) programs — used to import geometry for analysis and to set up parametric models in mechanical solutions from ANSYS — is essential for ANSYS DesignXplorer software to automatically perform numerous iterations in which various design geometries are created and analyzed. In this way, ANSYS DesignXplorer software is an effective means of integrating DFSS into a company's product development process. The software provides individual engineers a unified package for quickly performing probabilistic design and sensitivity analyses on thousands of design alternatives in a few hours; otherwise, this would take weeks of effort by separate statistics, simulation, DOE and CAD groups.

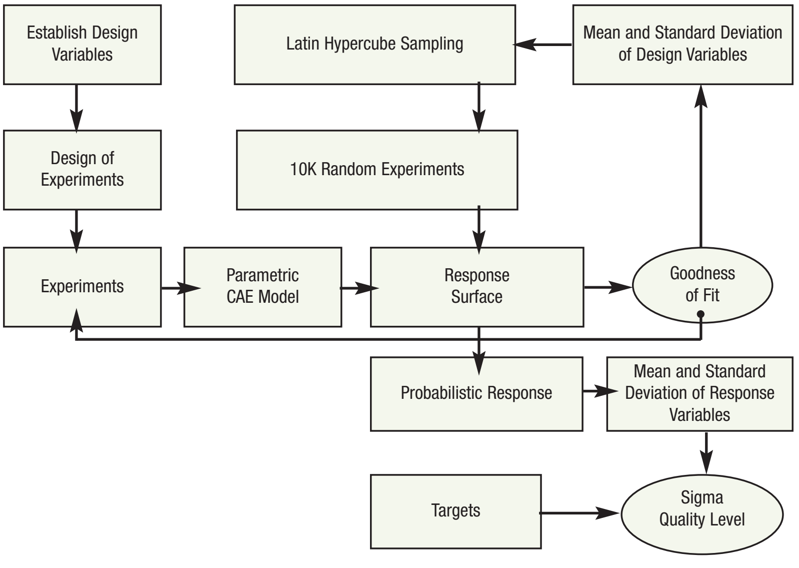
One recent project designed to improve hyper-elastic gasket configurations in proton-exchange membrane (PEM) fuel cells illustrates the value of the ANSYS DesignXplorer tool in DFSS applications. In this example, several gaskets provide a sealing barrier between the cell and approximately 200 bipolar cooling plates. In designing the fuel cells for commercial use in harsh environments, the goal was to lower the failure rate of the gaskets, which tended to leak on occasion — even in a carefully controlled research lab setting.



Probability density functions of parameters that vary with each analysis iteration



Scatter plots of analysis results were generated, along with bell-shaped probability density functions, in arriving at a robust gasket design.



With the workflow captured in the ANSYS Workbench platform, the process is highly repeatable and can be efficiently applied in optimizing the design of other gaskets.

First, design variables were established — gasket profile, gasket groove depth and the opposing plate's recessed pocket groove depth — that determined the overall compressive force of the gasket under a given bolt load. These were considered to be randomly varying parameters with given mean and standard deviations as determined through probability density functions generated by the ANSYS DesignXplorer tool. The software then was set up to automatically perform a series of DOE analyses in order to determine the gasket contact force for 10,000 different combinations of these variables. Variables were randomly selected by the software for each round of analysis using the Latin hypercube sampling technique.

Using ANSYS Mechanical analysis, solutions were arrived at in which (1) nonlinear capabilities characterize the hyper-elastic gasket material properties; (2) contact elements represent contact between the gasket and plates; and (3) parametric features automatically change the geometry of the gasket configuration for each of the 10,000 analyses.

Based on these analysis results, ANSYS DesignXplorer software generated a response surface of the contact force per unit length of the gasket in terms of probabilistic input variables. With the sensitivity established for each input variable on the contact force, scatter plots of the analysis results were generated along with bell-shaped probability density functions, which were compared to the upper and lower load limits of the fuel cell and cooler interfaces. Axial forces could not be so high as to break the plates, yet not so low as to cause leaking. From this data, the ANSYS DesignXplorer tool determined the sigma quality level based on the contact force target level.

The process succeeded in arriving at an optimal gasket shape that exceeded the sigma quality level, dropping the failure rate to an impressive three parts per million — a tremendous improvement over the 20 percent failure rate that the gaskets were experiencing previously.

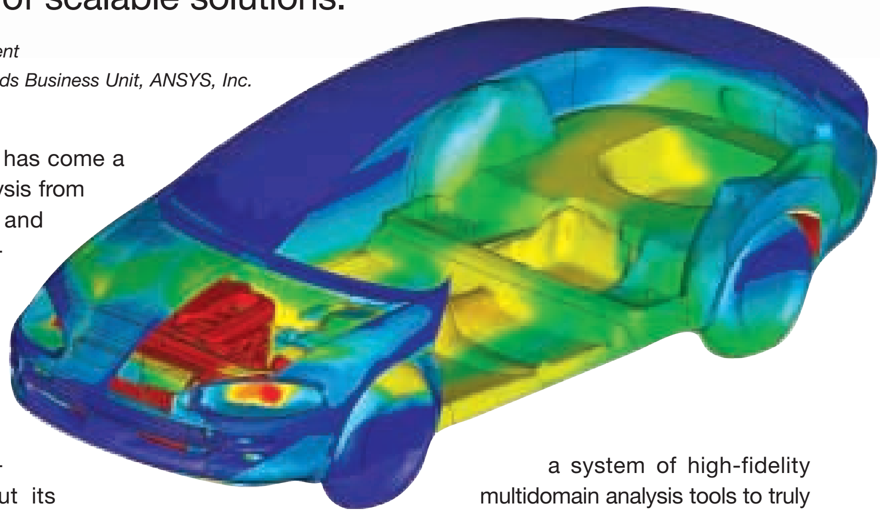
The entire process — including creation of the mesh models and completion of the 10,000 DOE analysis cycles — was completed in a matter of days by a single individual, as compared to months of effort that otherwise would have been required by separate design, statistics and analysis groups. Moreover, with the workflow captured in the ANSYS Workbench platform, the process now is highly repeatable and can be efficiently applied in optimizing the design of other gaskets merely by changing the CAD model and the upper/lower contact force limits. ■

More detailed information on the DFSS gasket project can be found in the ASME paper Fuel Cell 2006-97106 "Shape Optimization of Fuel Cell Molded-On Gaskets for Robust Sealing" by Vlahinos, Kelly, Mease and Stathopoulos from the International Conference on Fuel Cell Science, Engineering and Technology, Irvine, CA, June 19–21, 2006.

The New Wave of **Fluids Technology**

Fluid flow simulation software from ANSYS provides a broad range of scalable solutions.

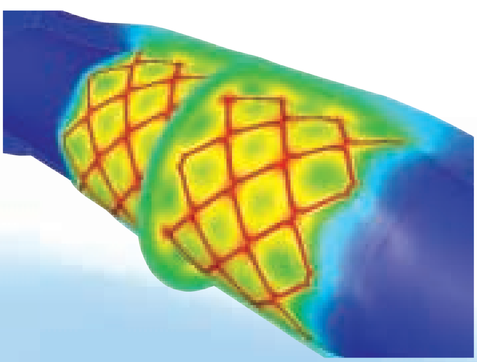
By Paul Galpin, Director of Product Management and André Bakker, Lead Product Manager, Fluids Business Unit, ANSYS, Inc.



Contours of temperature on a car body calculated in FLUENT software

The world of product engineering has come a long way in its quest to advance analysis from laborious hand-drawn sketches and simplistic models to virtual computer-created models initiated at the touch of a button. There has been a long evolutionary path from the inception of computational fluid dynamics (CFD) to today's integration of this technology into Simulation Driven Product Development (SDPD) processes. Throughout its history, ANSYS, Inc. has been a technological champion for such commercial engineering simulation. The company has viewed simulation as the key to predicting how products will perform; it has enabled the rapid comparison of many different alternatives prior to making a design decision — well before customers might identify problems. ANSYS now has a fluids product line that is both broad and deep, along with a large commercial and academic user base that is reaping the benefits.

This CFD evolution has required, and continues to demand, that ANSYS go beyond merely providing advanced mathematical flow solvers. ANSYS espouses a multiple physics approach to simulation in which fluid flow models integrate with other types of physics simulation technologies. The ANSYS vision is clear: to provide a system of high-fidelity multidomain analysis tools to truly enable SDPD.



Contours of drug concentration in a stent and capillary wall

SDPD is centered on the highly adaptive ANSYS Workbench architecture. The next major ANSYS Workbench release will provide another big step toward this vision. It will be the first release in which a number of the original Fluent CFD products will be data-integrated into the ANSYS Workbench platform, and thus the tools will work together with various other applications from ANSYS.

The ANSYS Workbench approach allows ANSYS to provide a large variety of software choices tailored to meet individual needs while ensuring interoperability and a clear future upgrade path. This includes a very broad fluids product line with all tools falling into one of three categories: general-purpose fluid flow analysis, rapid flow modeling and industry-specific products.

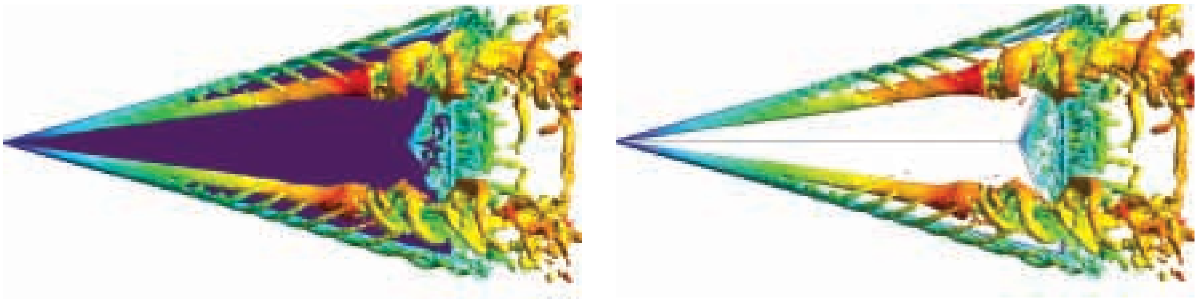
General-Purpose Fluids Solvers

The well-known FLUENT and ANSYS CFX products are the main general-purpose CFD tools from ANSYS. These two solvers, developed independently over decades, have a lot of things in common but also some significant differences. Both are control volume–based for high accuracy and rely heavily on a pressure-based solution technique for broad applicability. They differ mainly in the way they integrate the fluid flow equations and in their equation solution strategies.

The ANSYS CFX solver uses finite elements (cell vertex numerics) to discretize the domain, similar to those used in the mechanical analysis side of the business. In contrast, the FLUENT product uses finite volumes (cell-centered numerics). Ultimately, though, both approaches form "control volume" equations that ensure exact conservation of flow quantities, a vital property for accurate CFD simulations. ANSYS CFX software focuses on one approach to solve the governing equations of motion (coupled algebraic multigrid), while the FLUENT solver offers several solution approaches (density-, segregated- and coupled pressure-based methods). Both solvers contain a wealth of physical modeling capabilities to ensure that any fluids simulation has all of the modeling fidelity required.

The ANSYS CFX-Flo tool is a version of ANSYS CFX software that limits the physics accessible by the user to the models most commonly used by design engineers. It is compatible with other applicable ANSYS Workbench add-ins. The reduced complexity and cost of ANSYS CFX-Flo make it a good choice for design departments in organizations that already use ANSYS CFX software or other products compatible with the ANSYS Workbench environment.

The FLUENT for CATIA V5 product offers many of the same benefits of FloWizard and ANSYS CFX-Flo software. Completely embedded into the CATIA V5 system, it is fully compatible with the standard, full FLUENT solver. It is most useful for companies that use CATIA V5 in their design departments and FLUENT software in their analysis groups.



Turbulence on a delta wing calculated by ANSYS CFX software

These two core CFD solvers represent more than 1,000 person-years of research and development. This effort translates into the key benefits of fluid flow analysis software from ANSYS: experience, trust, depth and breadth. The fluids core solvers from ANSYS are trusted, used and relied upon by companies worldwide.

Rapid Flow Modeling

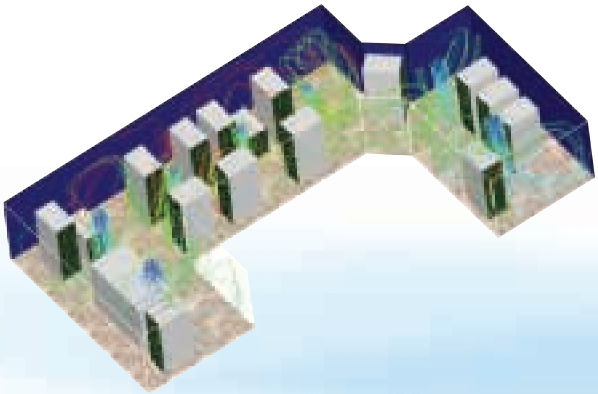
ANSYS addresses the fluid flow analysis needs of designers, who work on the front lines of their company's product development process and often need to make important design decisions quickly with no time to set up and solve complex mathematical models. For these time-limited engineers, ANSYS offers a choice of rapid flow modeling (RFM) products. RFM technology from ANSYS compresses the overall time it takes to do a fluid flow analysis by providing a high level of automation and focusing on only the most robust physical models. Three RFM tools are available: FloWizard, ANSYS CFX-Flo and FLUENT for CATIA® V5 software.

FloWizard software integrates all steps in the fluids process into one smooth interface. Computer-aided design (CAD) files can be sent to the FloWizard product, flow volumes extracted, models set up, calculations completed and HTML reports generated. FloWizard software is fully compatible with the FLUENT product, making it a good choice for designers in companies that use FLUENT technology in the analysis department.

Industry-Specific Fluids Simulation Tools

Flexibility and generality are important, but sometimes not required for specific applications. In addition to providing general-purpose CFD and rapid flow modeling products, ANSYS makes fluids simulation even more accessible and focused with its industry-specific analysis tools. These products are often called vertical applications because of the way they integrate all the steps for the analysis of a specific type of system into one package. The technologies offer industry-specific functions as well as employ the language of the industry in which they are used.

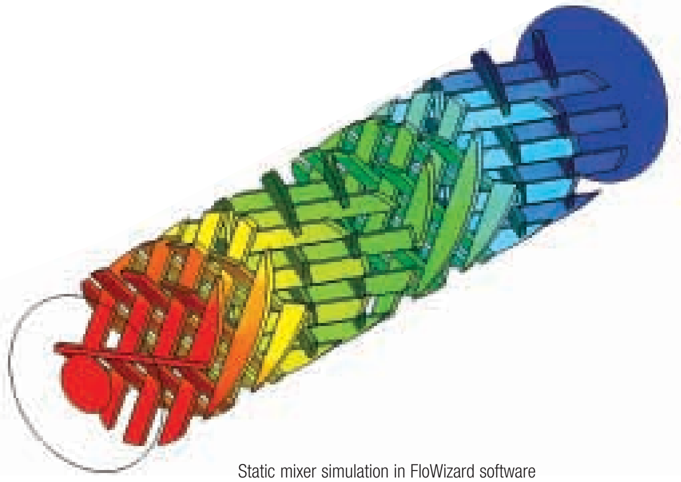
Turbomachinery is one of the world's single most successful CFD vertical applications, due to the similarity of the geometry and physics across a broad range of rotating



Airflow in a datacenter as simulated using ANSYS Airpak software machinery sectors. The turbosystem technology from ANSYS includes custom geometry and meshing tools as well as special modes within the general-purpose fluids simulation tools.

The ANSYS Icepak product is a family of applications focused on electronics design and packaging. In order to improve the performance and durability of electronic boards and other components for optimized cooling systems, the product calculates the flow field and temperatures in electronics and computer systems.

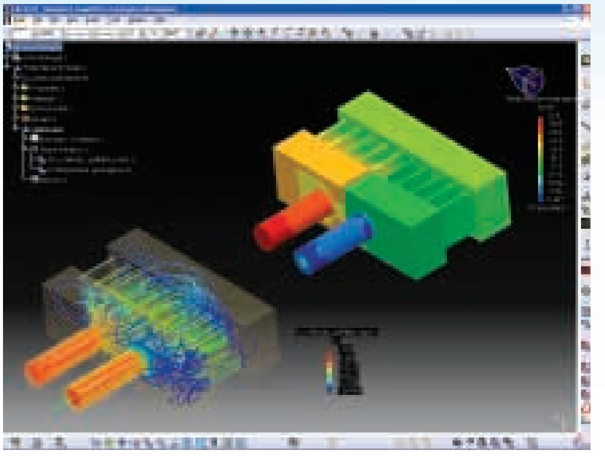
ANSYS POLYFLOW software is focused on the needs of the materials industry, such as polymer processing, extrusion, filmcasting and glass production. It can model the flow of fluids with very complex behavior, such as viscoelastic fluids. The ANSYS POLYFLOW product offers unique features such as the ability to perform reverse calculations to determine the required die shapes in extrusion. It also can calculate the final wall thickness in blow-molding and thermoforming processes.



Static mixer simulation in FloWizard software

The ANSYS Airpak product is aimed at the design of heating, ventilation and cooling systems in buildings, such as offices, factories, stadiums and other large public spaces. It accurately and easily models airflow, heat transfer, contaminant transport and thermal comfort in a ventilation system.

Finally, end-users can create their own vertical applications within the general-purpose fluids simulation products: ANSYS CFX software offers user-configurable setup wizards and expression language; FLUENT technology provides user-defined functions; and the FloWizard tool offers Python scripts. All of these can be used to create custom vertical applications. It is not uncommon for an analysis department to create such vertical applications for deployment within a design department. The main benefit of this approach is to ensure repeatable simulation process control, and hence quality control, for any CFD process.



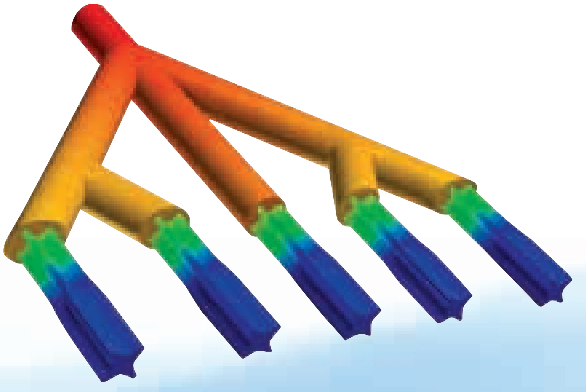
FLUENT for CATIA V5 software works within the CATIA V5 PLM environment, as shown in this simulation of a heat exchanger.

The Future of Fluids Simulation from ANSYS

To help customers replace more and more of their traditional capital-intensive design processes with a Simulation Driven Product Development method, ANSYS will continue to innovate and integrate.

In the very near future, users will see tremendous progress toward the ANSYS integration vision, including common geometry, meshing and post-processing tools for all users of CFD products from ANSYS. Many steps in the fluids simulation process will be automatically recorded, enabling parametric simulations. Improvements in fluid–solid connectivity will be evident, enabling a number of new multiphysics possibilities.

The upcoming ANSYS 12.0 release will lay a firm foundation for the future while carefully preserving and extending current software value. Over time, ANSYS plans to achieve the tightest possible integration of all its fluids technologies as well as an intimate integration with ANSYS mechanical technologies. The goal is to combine the best of the best into a simulation system with unprecedented power and flexibility. ■



The extrusion of a viscoelastic food material is simulated with ANSYS POLYFLOW software. The pressure drop between the inlet and the five outlets is shown. The outlet shape is computed as part of the analysis.

Multibody Dynamics: Rigid, Flexible and Everything in Between

Advances in simulation solutions for machine features accommodate more complex designs.

By Steve Pilz, Product Manager, ANSYS, Inc.

Simpler is better — that's what we've all been told. The more complicated something is, the more ways there are for it to break. This seems logical and is something we should consider as we invent new machines. The challenge is that simple machines do simple things and often can only do one thing well. A simple bottle opener, for instance, probably isn't the best tool for anything other than opening bottles, but it does what it was designed to do. Complicated machines — both mechanical and biological — have more parts, and often can be used to do more than one thing. As an example, the adult human body typically has 206 bones and can be used for all kinds of things from opening bottles to competing in triathlons. Inventing machines that can do a variety of things requires that the machines have multiple parts that work together, preferably without failing. Simulation tools in the product portfolio from ANSYS help make designing useful machines easier and faster, as well as more fun.

As simulation capabilities grow, an engineer's ability to simulate more complex machines increases.

Images:
motorcycle © iStockphoto.com/Paul Griffin bicycle © iStockphoto.com/Artsem Martysiok

Joints

When machines were simpler, there were fewer options, and multiple parts could be connected in mechanical software from ANSYS only using shared nodes, beam elements, coupling, constraint equations and node-to-node contact. These methods were adequate for many years, but eventually general surface contact was released to address the limitations. With this new functionality, parts undergoing large rotations, deformations, sticking, sliding and a host of other real-world behaviors could be modeled.

General surface contact became popular and widely used. It also became more robust and efficient with each successive ANSYS release for mechanical applications and is now considered mature, proven technology. One problem with the widespread use of general surface contact, however, is that sometimes it is more than is required. The relatively new capability to connect parts via joints has some potentially huge advantages that can be applied to many situations.

Joints were first released with the COMBIN7 element, which was used to model only pinned, or revolute, joints. At ANSYS 10.0, major advances to joint technology were made via the MPC184 element, which could be used to model multiple joint types, such as those that are translational, cylindrical, spherical, slot, universal, general or fixed. Joint elements are particularly interesting to those involved with the design of multiple-part machines because they can be used to enable large rotations and translations between parts at a very low computational expense. To illustrate the potential computational savings of using joints, a metal hinge is used as an example. (Figure 1.)

Joints: General Surface Contact vs. Revolute Joint Approach

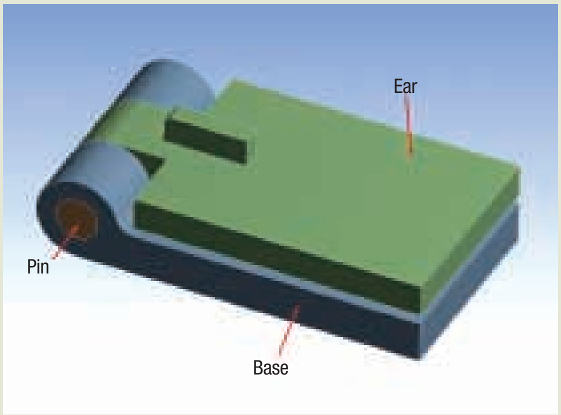


Figure 1. Hinge model

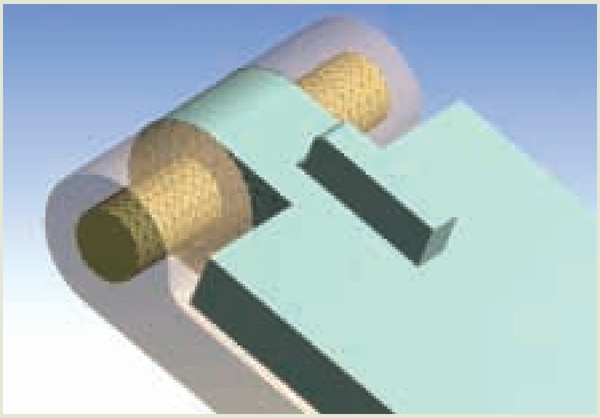


Figure 2. For this hinge model, general surface contact joints are used in three locations. First, where the ear meets the base, frictionless surfaces prevent translation along the axis of the pin and still allow rotation of the ear and base against each other at the joint. Second, bonded surfaces between the pin and the base prevent the pin from spinning or translating relative to the base. Then lastly, frictionless surfaces between the ear and the pin allow the ear to rotate freely about the pin.

There are many ways to set up a model for a metal hinge, but the two used in this investigation are a traditional general surface contact approach and a revolute joint approach (Figure 1). To simplify, the parts are set to be rigid so that problem size changes can be compared more easily. For each approach, a single CPU laptop is used to run the simulations.

In the general surface contact approach, to enable rotational freedoms but constrain all translationals except one, three contact surfaces are required (Figure 2) and one remote displacement, which rotates the hinge 90 degrees counter-clockwise. Using a few user-defined mesh specifications for surface contact size (body and edge sizing), the problem consisted of 7,188 elements (Figure 3) and took 2,249 seconds to solve.

By changing from a general surface contact approach to a revolute joint–based approach, there are three rigid parts and two joints connecting those parts to each other at the hinge: one revolute joint between the ear and the pin, and one fixed joint between the base and the pin. The pin could be suppressed since it won't perform any function once it is replaced with a revolute joint, but it is included in the model to make the run-time comparison equivalent with the general surface contact approach. The total problem size, as expected, is far smaller, uses only 14 elements (Figure 4) and requires a solution time of only 1.625 seconds.

So what have we learned? First, if detailed contact information at the hinge pin is unimportant, it is a lot more efficient to replace thousands of contact elements with a single revolute joint element. Doing that, the model can be solved in a fraction of the time it took to solve without the use of joints. Second, as can be seen from the element listing in Figure 4, even in a model in which contact surfaces are not specified, there are still contact elements — which come from use of the joint or MPC184 element — but far fewer of them.

TYPE	NUMBER	NAME
1	1	MASS21
2	1	MASS21
3	1	MASS21
4	1	CONTA176
5	1	TARGE170
6	180	CONTA174
8	1	TARGE170
9	178	CONTA174
10	180	CONTA174
11	178	TARGE170
12	576	CONTA174
13	1	CONTA176
14	1	TARGE170
15	832	CONTA174
16	1408	CONTA174
17	1408	TARGE170
18	288	CONTA174
19	832	CONTA174
20	288	CONTA174
21	832	TARGE170

Figure 3. Element description for hinge joint modeled with general surface contact

TYPE	NUMBER	NAME
1	1	MASS21
2	1	MASS21
3	1	MASS21
4	1	CONTA176
5	1	TARGE170
6	2	CONTA176
7	1	TARGE170
8	1	CONTA176
9	1	TARGE170
10	2	CONTA176
11	1	TARGE170
12	1	MPC184

Figure 4. Element description for hinge joint modeled with a revolute joint

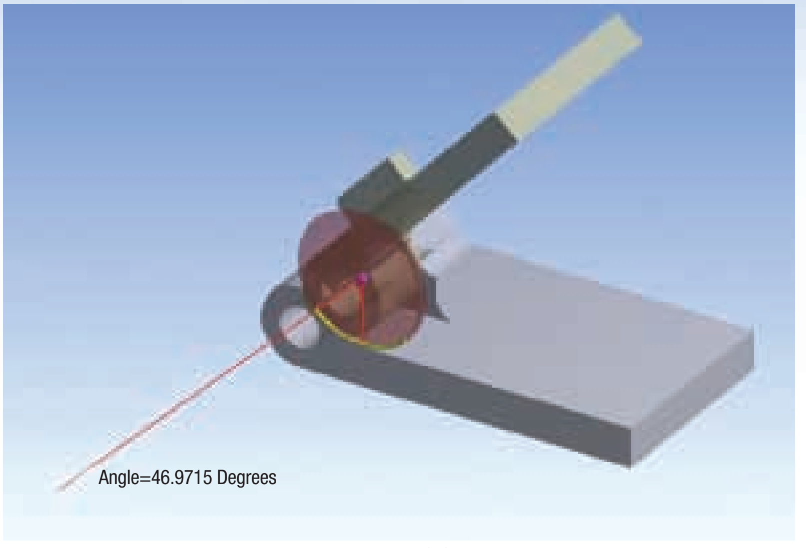


Figure 5. Interactive joint manipulation is possible within the ANSYS Rigid Dynamics module, performed on a computer screen by using the mouse to move the model.

Interactive joint manipulation allows the user to solve a model essentially in real time — the explicit solver produces a kinematic solution with part positions and velocities — using the mouse to displace the parts of the model. This tool is on the menu bar in the Connections folder. New Configure, Set and Revert buttons can be used to exercise a model that is connected via joints, set a configuration to use as a starting point or revert back to the original configuration as needed. In the case shown in Figure 5, before finding a solution, the hinge has been rotated a little more than 46 degrees to verify that the joint is, in fact, behaving like a hinge.

The ANSYS Rigid Dynamics module is run using the same techniques that are used in ANSYS Workbench *Simulation* — attaching to the CAD or the ANSYS DesignModeler model, using the model tree, populating the Connections folder and inserting New Analysis, for example.

The combination of the explicit Runge–Kutta time integration scheme and a dedicated rigid body formulation creates a product that while limited to working only with completely rigid parts,

ANSYS Rigid Dynamics

The ANSYS Rigid Dynamics module, first released at Version 11.0, makes extensive use of joints for connecting parts. This is an ANSYS Workbench add-on tool for users who have ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics licenses. The module enhances the capability of those products by adding an explicit solver that is tuned for solving purely rigid assemblies. As a result, it is significantly faster than the implicit solver for purely rigid transient dynamic simulations. The ANSYS Rigid Dynamics module also has added interactive joint manipulation and ANSYS Workbench *Simulation* interface options.

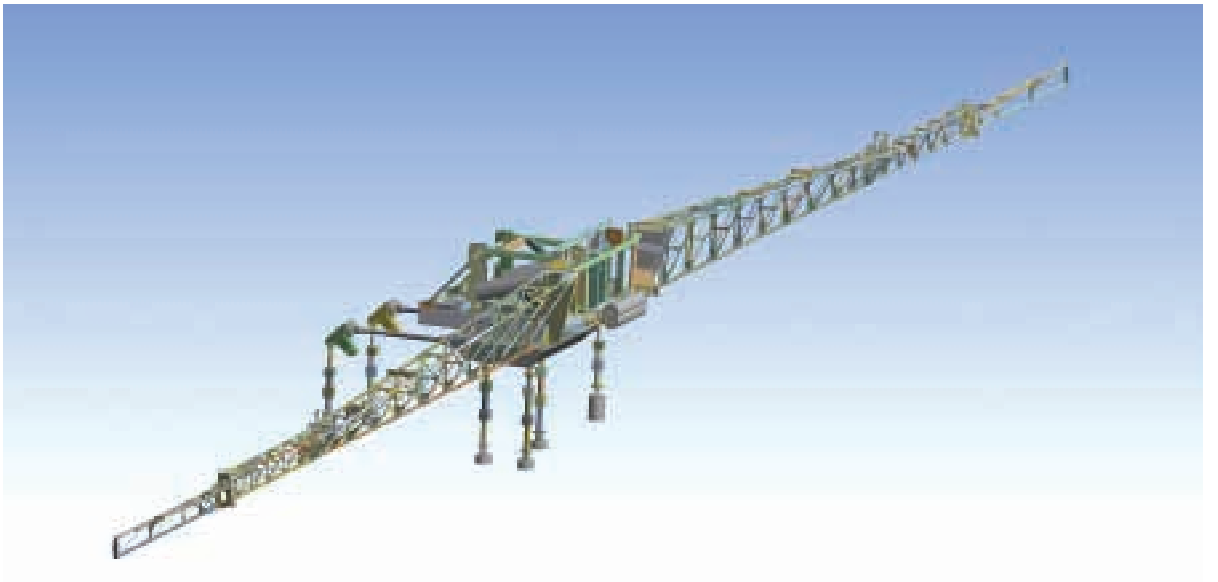


Figure 6. Folding arms of John Deere agricultural sprayer model to be subjected to time–history loading

Image courtesy Brenden L. Stephens, John Deere

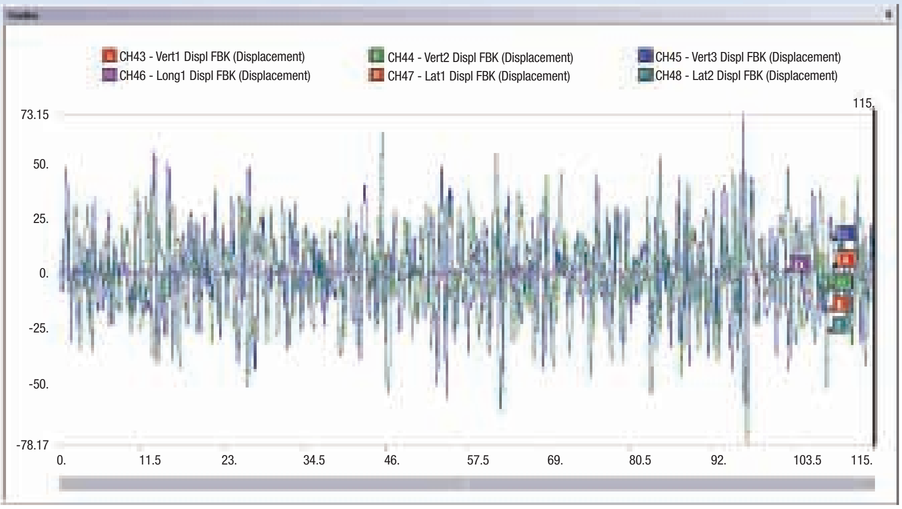


Figure 7. Time–history loading at six different geometric locations along the sprayer model in Figure 6
Image courtesy of Brenden L. Stephens, John Deere

is extremely well suited to solving multi-jointed assemblies, such as the folding arm agricultural assembly (Figure 6). This scheme is adept at handling complex time–history input (Figure 7) and is extremely fast compared to more traditional solvers. Solve time, even for complex assemblies, is typically measured in seconds and minutes rather than in hours and days. One caveat worth mentioning is that, at release 11.0, parts need to be connected with joints rather than contact when using the ANSYS Rigid Dynamics capability. If contact is required to accurately represent the part interactions, flexible dynamics simulation is required.

The ANSYS Rigid Dynamics tool should be used first on any complex, multi–part assembly with connections. Fast solution times can help users quickly find joint definition problems, inadequate boundary conditions, over-constraints and other problems. With the time saved, multiple design ideas can be analyzed in the same amount of time that it previously would have taken to simulate a single concept.

ANSYS Flexible Dynamics

Is the ANSYS Rigid Dynamics tool all that is needed to fully understand a prototype of a machine? What happens if the parts deform? Will they break? Will they fatigue and fail after a short time or only after extreme use? If parts bend, twist and flex, will the machine still perform its intended function?

The ANSYS Rigid Dynamics capability, for all its strengths, doesn't provide a complete picture of a machine's performance. In a thorough machine prototype investigation, the next step is a flexible dynamics analysis, which allows some or all of the machine's parts to behave as they would in the real world — flexing, twisting and deforming. Flexible dynamics allows users to examine parts to identify whether they are stiff and light, as they would be if made from titanium, or heavy and flexible, as they would be if made from rubber.

A more in-depth explanation of the use of ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics products running flexible nonlinear dynamics simulations is necessary to demonstrate the steps required to take an all-rigid dynamics model and turn it into a partially or completely flexible model. This translation from a rigid to a flexible model includes material assignment, meshing and solver setup. Without writing a spoiler to any future articles on this subject, this is remarkably easy to do.

The simple machines have already been invented. We don't really need a more efficient bottle opener. With the addition of more realistic and faster modeling solutions — achieved by combining the ANSYS Rigid Dynamics module and ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics software — complicated machines can be less prone to failure and produce fewer career-limiting disasters. ■

Nonlinear Simulation Provides More Realistic Results

When parts interact and experience large deflections and extreme conditions, nonlinear technology is required to simulate real-life situations.

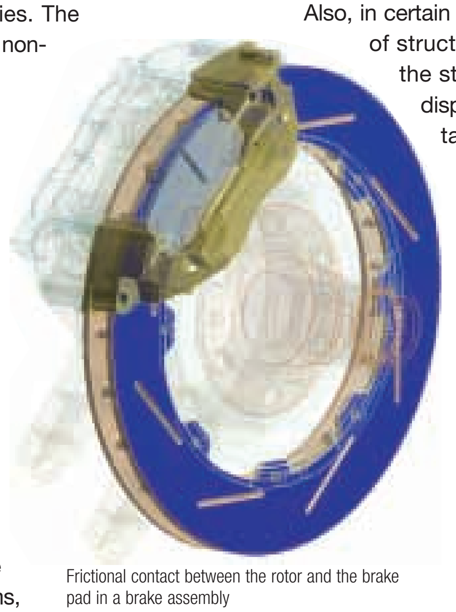
By Siddharth Shah, Product Manager, ANSYS, Inc.

In today's competitive environment in which everyone strives to develop the best design with the best performance, durability and reliability, it is unrealistic to rely on linear analysis alone. Analyses must be scaled from single parts and simplified assembly-level models to complete system-level models that involve multiple complex subassemblies. As more parts get added to a simulation model, it becomes more difficult to ignore the nonlinear aspects of the physics, and, at the same time, expect realistic answers.

In some situations involving single- or multiple-part models, analysis with linear assumptions can be sufficient. However, for every assumption made, there is some sacrifice in the accuracy of the simulation. Ignoring nonlinearities in a model might lead to overly conservative or weak design in certain situations, or might result in the omission of unexpected but valuable information about the design or performance of the model. It is essential to understand when and when not to account for nonlinearities. The following are some situations in which nonlinearities are commonly encountered.

Contact

Currently, auto-contact detection in ANSYS Workbench *Simulation* allows users to quickly set up contact (part interactions) between multiple entity types (solids, sheets, beams). However, in cases in which two parts interact with each other, the parts might stick or slide against each other instead of remaining static. Also, their stiffness might change depending upon whether they touch each other or not, as is seen with interference or snap-fit cases. Ignoring sliding may be acceptable for a large class of problems, but for those with moving parts or that involve friction, it is unwise to make this assumption.



Frictional contact between the rotor and the brake pad in a brake assembly

Geometry

In certain situations, the deflections of a structure may be large compared to its physical dimensions. This usually results in a variation in the location and distribution of loads for that structure. For example, consider a fishing pole being bent or a large tower experiencing wind loads. The loading conditions over the entire body of the structure will change as the structure deflects.

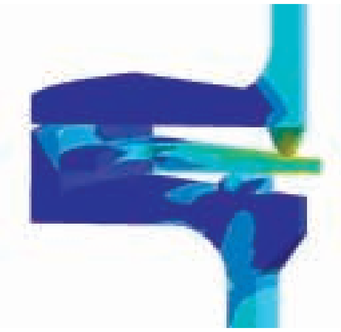
Top-loading simulation of a plastic bottle

Also, in certain slender types of structures, membrane stresses may cause the structure to stiffen and, hence, reduce displacements. One example of this is fuel tanks used for satellite launchers and spacecraft. If accurate displacements are to be computed, geometry nonlinearities have to be considered.

Material

Material factors become increasingly important when a structure is required to function consistently and reliably in extreme environments — such as structures that must operate at high temperatures and pressures, provide earthquake resistance, or be impact-worthy or crash-worthy. Plastics, elastomers and composites are being used as structural materials with increasing frequency. These materials do not follow the linear elastic assumption of stress–strain relationships. Structures made with these materials may undergo appreciable changes in geometric shape before failure. Without accounting for this material behavior, it can be impossible to extract meaningful and accurate information from their simulations.

In the past, nonlinear analysis was associated with heavy investment in training, resources and manpower. For many, the software appeared cumbersome, challenging and intimidating. It was acceptable and often preferable to get by with physical testing alone. That is not the case today, however. Nonlinear structural simulation is no longer an intimidating tool, but rather one that ANSYS has made available to all engineers by fusing its complex physics into an easy-to-use interface in the ANSYS Workbench environment. ■


Medical check valve

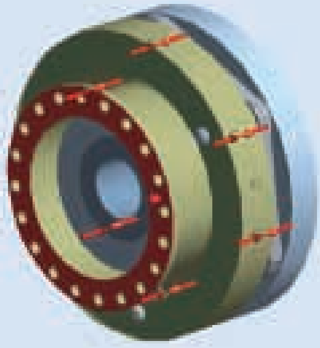
Turbomachinery Coupling

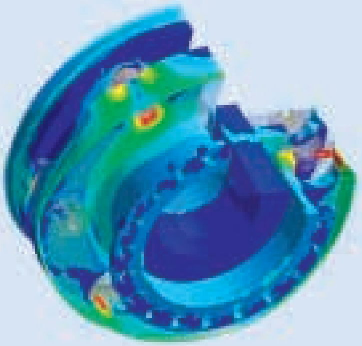
Bibby Transmissions Group — a long-time ANSYS DesignSpace user — has been a world leader for many years in the design and manufacture of couplings for use in industrial markets. The company's high-speed disc couplings, designed by its TurboFlex division, have been a popular choice for transmission couplings among the power, chemical, steel and water treatment industries.

Engineers at Bibby found that with linear analysis assumptions, material yielding occurred around clearance holes where the flexible coupling was mounted and also when the coupling was rotating near its operating speed. Knowing that they were not capturing material behaviors related to contact and preloading conditions, engineers at Bibby felt a need to model the material plasticity and calculate plastic strains and deflection. This analysis was undertaken to ensure that the loading-induced plasticity was localized and did not induce global failure for the coupling. The simulation required nonlinear modeling of contact in which the couplings used an interference fit, material behavior for the hub and spacer, and bolt preloading for the couplings.

Bibby engineers successfully set up this model within the ANSYS Workbench *Simulation* tool using the previously mentioned nonlinearities and were able to accurately predict the observed behavior. In addition, they were able to identify operating speed — not torque as had been previously believed — as the dominant factor that influenced the observed plastic deformation. This valuable information could not have been obtained by physical testing alone.

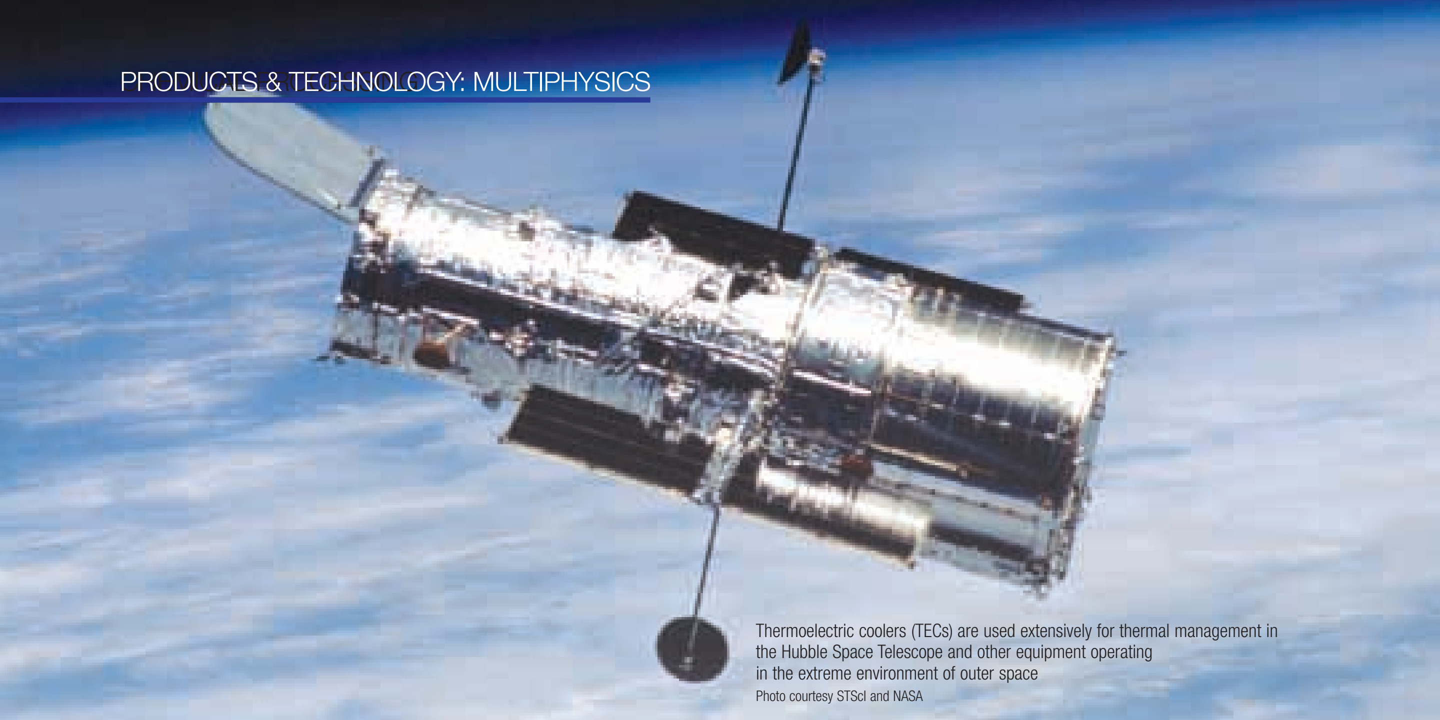
Thanks to Wilde FEA Ltd. for assistance with this article.


The assembly model includes pretension bolted joints and BISO material for the hub and spacer.


The von Mises stress exceeds the yield limit of 700 MPA, and yet it is localized.

"Through the ANSYS Workbench platform, we have a tool that allows us to increase the performance of our products. Drastic reductions in weights and inertia of the couplings have been achieved without compromising the strength of the unit. Lateral vibration of couplings is now being estimated to a level of confidence previously unattainable without days of computation and cost."

— Ron Cooper, Technical Director
Bibby Transmission, U.K.



Thermoelectric coolers (TECs) are used extensively for thermal management in the Hubble Space Telescope and other equipment operating in the extreme environment of outer space
Photo courtesy STScI and NASA

High Performance from **Multiphysics Coupled Simulation**

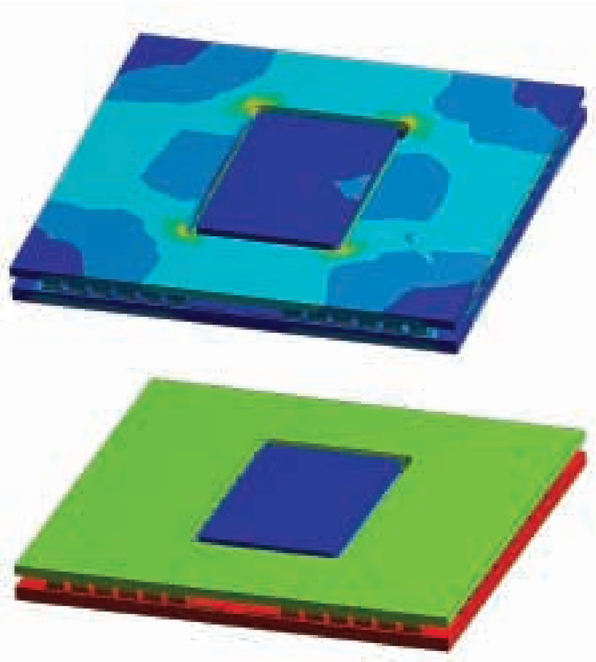
Engineers use ANSYS Multiphysics to study the mechanical strength and thermal performance of an innovative thermoelectric cooler design.

By Robin McCarty, Senior Engineer for Product and Process R&D, Marlow Industries, Texas, U.S.A.

Thermoelectric coolers (TECs) serve as small heat pumps, utilizing semiconductors for the cooling action in an enclosed package without any moving parts. Because of their quiet operation and small size, the devices are used extensively for spot-cooling electronics in aerospace, defense, medical, commercial, industrial and telecommunications equipment. In the extreme environments found in satellites and space telescopes applications, TECs often are stacked on top of one another to achieve the required cold-side temperatures. The traditional multistage configuration is pyramidal in shape, with the unavoidably tall profile posing packaging problems in applications with limited vertical space.

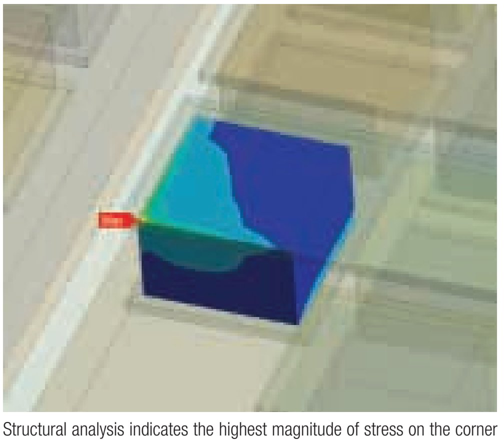
To address these issues, Marlow Industries developed an innovative new planar multistage TEC (patent pending) that reduces overall device height by arranging the thermoelectric elements side-by-side in a single plane, instead of stacking them. Because this configuration radically changed the structure, engineers used ANSYS Multiphysics software in evaluating the thermoelectric (TE) performance and thermomechanical stresses of the device, enabling the company to meet critical deadlines for launching the new product in a competitive market.

Thermoelectric coolers serve as small heat pumps, utilizing semiconductors for the cooling action in an enclosed package without any moving parts.



Coupled-physics simulation determined the temperature distribution throughout the device (top). These results were applied to the TEC assembly to perform a static structural analysis of the structure (bottom).

The company selected the ANSYS Multiphysics product because it is recognized as the only commercial finite element analysis package with the capability to model 3-D thermoelectric effects with the required level of accuracy. Given the multiphysics capabilities of the software, a fully coupled thermoelectric simulation could be performed, calculating the current densities and temperatures in the TEC considering both Joule heating and the Peltier effect. Marlow engineers used the calculated temperatures from the thermoelectric analysis of the TECs to perform a static structural analysis, which then was used to predict thermal stresses in the thermoelectric materials due to temperature differences in the TEC assembly.



Structural analysis indicates the highest magnitude of stress on the corner of the thermoelectric element where Marlow has historically seen cracking.

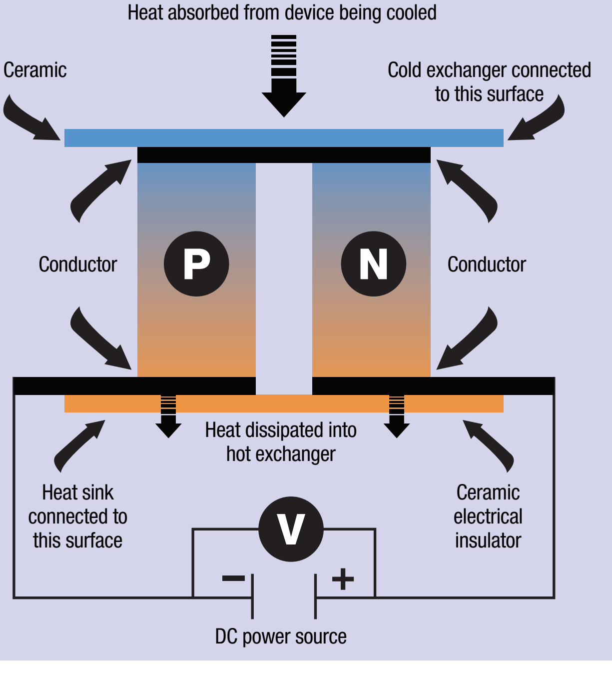
The objective of the thermoelectric simulation was to determine temperature distribution throughout the device. For creating the analysis model, a constant temperature condition was applied to the bottom of the mounting solder, and a radiation boundary condition was applied to the cold-side ceramic. A heat load (simulating the heat-producing device to be cooled) was applied to the cold side of the TEC, and a DC current was applied to the TEC's electrical terminals to drive the thermoelectric cooling. From this coupled-physics simulation, the minimum cold-side temperature, temperature uniformity of the top stage, voltage drop and electrical resistance of the TEC were determined.

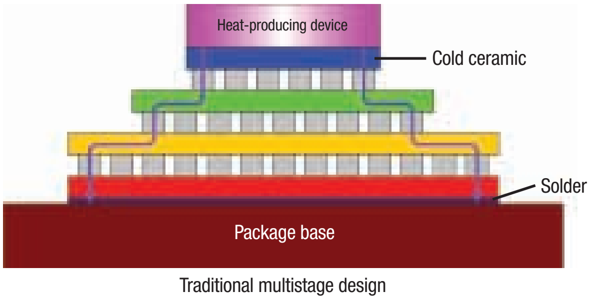
Once the temperature distribution of the TEC assembly was calculated from the thermoelectric model, it was applied to the TEC assembly in a static structural analysis. To mimic the TEC's mounting conditions, the solder on the

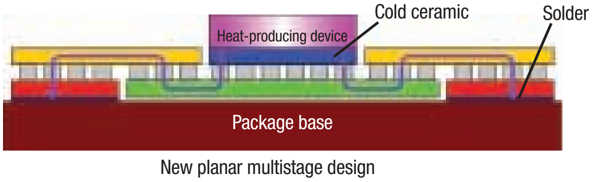
How a Thermoelectric Cooler Works

A thermoelectric cooler operates based on a principle known as the Peltier effect, in which cooling occurs when a small electric current passes through the junction of two dissimilar thermoelectric materials: a "p-type" positive semiconductor with a scarcity of electrons in its atoms and an "n-type" negative semiconductor with an abundance of electrons. Current is carried by conductors connected to the semiconductors, with heat exchanged through a set of ceramic plates that sandwich the materials together.

When a small positive DC voltage is applied to the n-type thermo element, electrons pass from the p- to the n-type material, and the cold-side temperature decreases as heat is absorbed. The heat absorption (cooling) is proportional to the current and the number of thermoelectric couples. This heat is transferred to the hot side of the cooler, at which point it is dissipated into the heat sink and surrounding environment.







In contrast to traditional multistage thermoelectric coolers with elements stacked in a pyramid shape (left), the new Marlow flat configuration (right, patent pending) arranges stages side by side. The new design reduces the height of the device and also changes heat flow through the ceramic material (denoted by the purple arrow).

hot side of the device was fixed on the bottom surface. Maximum principal stress was used to evaluate and compare the TEC designs because it can be directly related to the failure of a brittle material, such as bismuth telluride.

The testing team identified the thermoelectric element with the maximum stress and then refined the finite element mesh in that area to ensure that stress convergence had been obtained for the structural simulation. Using a plot of maximum principal stress distribution in a typical TE element, the engineering team found that the maximum stress occurs on the corner of the TE element, which correlated to where Marlow historically had seen cracking in thermoelectric elements that resulted in device failure.

To validate the new planar multistage designs, Marlow evaluated the mechanical stress levels for a thermoelectrically equivalent traditional multistage device and a planar multistage device. Each device consisted of three stages equivalent with thermoelectric element dimensions and thermoelectric element count per stage. In the model, three different currents were evaluated, and the maximum principal stress located in the most highly stressed thermoelectric element was noted.

Through these analyses, Marlow configured planar designs with maximum principal stress levels comparable to the traditional multistage devices. Thermal performance also was nearly equivalent. The correlation between the stress results for the traditional multistage and planar multistage devices provided confidence in the new planar multistage design concepts. This type of evaluation would not have been possible without the multiphysics simulation capabilities available in software from ANSYS. ■