SPOTLIGHT ON ANSOFT
PAGE 19
Simplify Cluster Deployment and Start Working Sooner

Intel and SGI Solutions for ANSYS

Innovative SGI clusters, based on the Quad-Core Intel® Xeon® processor, work together to control complexities, even in the most challenging CAE environments. Certified as Intel Cluster Ready, these systems can help you increase productivity and reduce the effort to configure, acquire, and deploy ANSYS and FLUENT.

**Intel Cluster Ready**

The Intel® Cluster Ready program makes it easy to acquire and use clusters built with Intel components. Applications that have been tested to run on clusters certified as Intel Cluster Ready will run on any other Intel Cluster Ready certified cluster, freeing you to focus on getting your job done.

**Learn more**

Find out about how SGI and Intel are teaming up to meet the demanding requirements of high performance computing and to ensure that ANSYS and FLUENT run optimally on the SGI platform.

Learn more about the Intel Cluster Ready program at:

www.intel.com/go/cluster
A Big Step Forward in Mechatronics

The acquisition of Ansoft expands the breadth of multiphysics capabilities from ANSYS and gives engineers a powerful range of simulation tools for systems, blending together mechanical and electronics designs.

Mechatronics-based products combine mechanical assemblies with electronics, intelligent control systems, electromagnetics and electromechanical components. They are all around us — and growing exponentially. Electronics-based hand-held products, unheard of years ago, are now commonplace, and traditionally all-mechanical products such as cars, planes, toys and appliances now have increasing levels of electronic circuitry.

One formidable barrier in developing these mechatronics systems is that mechanical and electronics development processes are usually not performed in an integrated manner. Mechanical computer-aided design (MCAD) and electronic design automation (EDA) systems typically are incompatible and do not exchange data smoothly. These different disciplines often work independently — risking problems downstream when subsystems are pieced together, resulting in missed opportunities to collaborate in optimizing the system’s overall multiphysics performance. In development of a stealth fighter aircraft, for example, designing the exterior shape of the plane to minimize reflection of electromagnetic waves from ground radar may not result in optimal aerodynamics and might limit options for designing the landing gear or weapon delivery systems.

A big step forward in bringing together these otherwise separate physics was taken recently with the ANSYS, Inc. acquisition of Ansoft Corporation — a provider of EDA software and simulation tools for electromagnetics, electromechanical, circuit and electronic systems. For more on this important development, see the industry spotlight section in this issue for a history of Ansoft, a lineup of software products, the synergy of the two companies’ solutions, and case studies showing how companies put these technologies to work.

This entry of ANSYS into the electronic design software industry broadens the company’s range of simulation solutions. Furthermore, adding Ansoft electromagnetics and electromechanical functionality to ANSYS technologies for structural and fluid dynamics simulations greatly expands the range of multiphysics simulations that can be performed — especially those involving mechatronics.

With these comprehensive multiphysics solutions, electronics engineers can readily evaluate stresses in semiconductor packages and printed circuit boards as well as assess reliability of hand-held devices undergoing shock and vibration. They can study various cooling strategies for electronics and better understand heat flow in high-density packages. All of this can be accomplished while also optimizing the design of radio-frequency and microwave components or studying signal and power integrity of high-performance electronics. The various simulations can be tied together within the ANSYS Workbench framework for a smooth exchange of data between various field solvers and design tools; the information can be linked with ANSYS Engineering Knowledge Manager (EKM) software for managing simulation data.

Such a unified approach, breadth of engineering solutions and depth of multiphysics technologies gives development teams the tools they need in a competitive environment, where the ability to design mechatronics-based systems better and faster will likely be decisive for a growing number of companies in the coming years.

John Krouse, Senior Editor and Industry Analyst
FEATURES

4 AUTOMOTIVE
First to the Finish Line
In a sport where winning is often decided by split seconds, the BMW Sauber F1 Team uses fluid dynamics solutions from ANSYS to lower lap times through improved vehicle aerodynamics.

8 INDUSTRIAL EQUIPMENT
New Check Valves Swing into Action
ANSYS multiphysics simulation and design optimization determine petrochemical check valve flow characteristics overnight — a task that otherwise could take years to complete.

11 ANALYSIS TOOLS
Making an Impact
Modern explicit solutions enable the study of blast or high-impact scenarios.

15 THOUGHT LEADER
Multiphysics Maestro
An industry visionary shares insights into the evolution of multiphysics solutions and future challenges to overcome.

SPOTLIGHT ON ANSOFT

20 A History of Innovation
Ansoft has a long history of developing leading simulation solutions for the electronics industry.

23 ANSYS and Ansoft: The Power of Synergy
Integrating Ansoft tools with technologies from ANSYS combines the best of both worlds for developing electronic products.

26 Aligned with the ANSYS Vision
The Ansoft product suite will help deliver benefits to the entire ANSYS engineering simulation community.

27 Electric Motors Advanced by “Ultra” Power Storage
Electromechanical simulation tools aid in the design flow of hybrid–electric systems.

29 Microwave Simulation, Macro Benefits
Electromagnetic simulation finds applications in high-performance antenna systems and electronics.

31 Impeding Interference
Panasonic improves signal integrity design for a remote surveillance camera using electronics design software from Ansoft.
SIMULATION @ WORK

33 AUTOMOTIVE
Decreasing the Shock in Shock Absorbers
Engineering simulation improves valve design for an automotive shock absorber.

34 ENERGY
Scheduling Replacements Smartly
Simulation is used to effectively predict crack growth that could lead to power plant valve failure.

36 TURBOMACHINERY
What’s Shakin’?
The combination of 3-D structural dynamics, ANSYS Workbench and classical rotordynamics modeling techniques helps solve rotating machinery vibration problems.

38 PROCESS INDUSTRIES
Designing Safe Crackers
Fluid and structural simulation combine to help researchers analyze a variety of stresses on catalytic cracking equipment.

DEPARTMENTS

40 ANALYSIS TOOLS
Multibody Dynamics: Rigid and Flexible Methods
Choosing the right simulation method for dynamic assemblies doesn’t have to be risky.

43 ANALYSIS TOOLS
Extensive Multiphase Flow Capabilities
Fluid dynamics simulation provides a wide range of multiphase flow capabilities to meet challenging industrial needs.

46 TIPS AND TRICKS
Analyzing Viscoelastic Materials
Mechanical solutions from ANSYS have convenient tools for calculating deformation of materials in which stiffness changes as a function of loading, time and temperature.

48 ACADEMIC
Small Bubbles, Large Benefit
Air injection simulations show promise for reducing ship drag by injecting air bubbles to reduce skin friction along the hull of a vessel.

49 OUTSIDE THE BOX
Snowflake by Snowflake
Simulation provides insight about snow safety.

For ANSYS, Inc. sales information, call 1.866.267.9724, or visit www.ansys.com.
For address changes, contact AdvantageAddressChange@ansys.com.
To subscribe to ANSYS Advantage, go to www.ansys.com/subscribe.
First to the Finish Line

In a sport where winning is often decided by split seconds, the BMW Sauber F1 Team uses fluid dynamics solutions from ANSYS to lower lap times through improved vehicle aerodynamics.

By John Krouse, Senior Editor and Industry Analyst, ANSYS Advantage

Formula 1 (F1) race cars are strange-looking beasts, resembling Star Wars fighting machines more than motor vehicles. The numerous odd-shaped appendages, exposed tires and open cockpit contribute to considerable air resistance. Indeed, the aerodynamic drag of these cars is worse than that of a brick.

The reason for such low aerodynamics is that F1 rules mandate these configurations — along with standardized tires and restrictions on engine development — to limit car speeds, thereby making for a slower, safer and more exciting race where vehicles are more evenly matched.

Nevertheless, within these evolving restrictions, aerodynamics is a key performance driver for teams wanting to improve car lap times by the fractions of a second needed to win races, according to Willem Toet, head of aerodynamics for the BMW Sauber F1 Team.

Given these demands, developing optimal racecar aerodynamics is a huge engineering challenge involving a wide range of conflicting requirements. In straightaways, highest speeds at full throttle are gained with aerodynamic "slippery" vehicles, in which components exposed to the air stream are integrated. These include the chassis, underbody, engine cover with air inlets, vehicle nose, side pods with cooling inlets, wheels, brakes, suspension and exhaust system.

During cornering, braking, accelerating and high-speed maneuvering in the pack, the key is road-holding ability from downward forces produced by airfoil surfaces on the vehicle. At speeds above 170 km/h (106 mph), this force against the track typically equals total vehicle weight so that the car could drive upside down across the ceiling — at least in theory.

These high downward forces inevitably increase air resistance and lower the vehicle's top speed, however. The aerodynamicist must balance...
these trade-offs so that the time gained on the bends is not lost on the straight-aways. Every part of the bodywork must be designed to provide maximum downward force with a minimum of drag. Moreover, downward force at the front and rear of the vehicle must be carefully apportioned depending on characteristics of individual racetracks. Also, aerodynamic parts such as fins, diffusers, wings and barge boards must provide optimal downward force without interfering with air inlets or engine cooling. The aerodynamic design from the front of the car to the rear depends on the shape and placement of all these components, and the vehicle reacts extremely sensitively to the slightest design alterations.

Compounding these technical complexities, teams have only nine months to develop a new car. Chassis, engine, transmission and, above all, aerodynamic concept must be right at the very first attempt. There is no time for multiple prototype cycles, and, when the first tests are held in January, it is generally too late for wholesale changes. Furthermore, during the racing season the idea of “standing still” with the same design is an alien concept. “At practically every one of the 18 races, teams make some kind of improvement so that their cars at the end of the season have very little in common with the original designs,” said Toet. “With a race scheduled every two weeks, the clock is ticking to make performance-enhancing modifications on their cars before competitors do.”

Simulation is a Must-Have Tool

In contending with the above technical challenges and demanding timetables, today’s racing aero-dynamicists simply cannot rely on intuition or trial-and-error methods. Rather, engineering simulation has become a standard part of vehicle development for most teams.

Finite element analysis became universally used in the racing industry as early as the 1980s, while simulations of air flow appeared in the early to mid-1990s. In recent years, computational fluid dynamics (CFD) simulation has experienced a real boom period in racecar design because of the ability to quickly and cost-effectively study aerodynamic efficiency and investigate the impact of design modification and alternative what-if changes on vehicle performance.

Fluid flow simulations are used extensively in these applications and range from the stationary analysis of individual components (wing profiles, for example) up to investigations of the entire vehicle as well as non-stationary simulations, such as interactions within the vehicle when overtaking another car. Also covered are fluid flow considerations that are outside the field of aerodynamics, such as sloshing inside a fuel tank.

Demonstrating a commitment to expanding its use of CFD, the BMW Sauber F1 Team recently upgraded its high-performance supercomputer cluster specifically developed for efficiently processing these simulations. Made by DALCO AG of Switzerland, the cluster has a peak computing speed in excess of 50 teraflops (one trillion calculations per second) with a compute section based on Intel® Xeon® 5160 dual-core and E5472 quad-core processors. With this processing power and the efficiency of fluid dynamics solutions from ANSYS, BMW Sauber F1 Team’s engineers can process full-vehicle simulations in a matter of a few hours, instead of the weeks otherwise required on conventional machines.
"The big difference with CFD compared to wind tunnels is that you not only get results, but you also get an understanding of what goes on. Wind tunnel testing remains important with experimental work and CFD complementing each other," said Theissen. He noted that comparative wind tunnel measurements are also used for calibration and validation of the fluid flow calculations to increase the accuracy and reliability of simulation results. The range and detail of data collected, reproducibility of the results, and a better understanding of complex aerodynamic interactions are all strong arguments in favor of CFD simulations.

In an expanding range of studies, simulation is used to overcome limitations of wind tunnel testing. Because vehicles generally are stationary for wind tunnel tests, for example, evidence concerning air-flow characteristics can be rather vague in many regions. Also, wind tunnel tests are of little use in investigating heating and cooling effects because the engine is not running and brakes are not at operating temperature. In contrast, fluid dynamics simulations take all these factors into account, and calculations can be applied to all physical parameters, including those

Fluid Dynamics Complements Wind Tunnels

"The latest upgrade of our supercomputer was a decisive reinforcement of our CFD capacity. Unlike other teams, we didn’t plan to build a second wind tunnel. Instead, we have used the key relationship commitment with our high-performance computing (HPC) partners, including ANSYS, to continue to develop and exploit the expanding potential for CFD that high-performance computing gives us," explained Mario Theissen, BMW Motorsport director.

He added that wind tunnel testing will continue as an important design element of their Formula 1 racing car design. The BMW Sauber F1 Team’s wind tunnel generates wind speeds up to 300 km/h (188 mph) and features what is known as a “rolling road” that can simulate the interaction between the vehicle and the road surface. Using the wind tunnel, engineers can readily see the effect, on the actual vehicle, of minor adjustments made on the spot to part geometries and orientations.
involving variations in multiple inter-related parameters. In the final analysis, however, it is the testing on the racetrack itself that is the actual yardstick for evaluating the success of the aerodynamic methods in use and deciding to what extent they have been effective.

Coping with New Regulations

Beginning with the 2009 racing season, new regulations for the configuration of Formula 1 cars will drastically limit the use of aerodynamic surfaces. Many components, such as winglets to direct airflow, will become history. These constraints will certainly result in slower lap times, and performance of all the racing teams will likely become more closely matched.

“New regulatory changes do not mean that CFD simulations will become less important or that the work of aerodynamicists will diminish,” Toet emphasized. “On the contrary, demands of the new motor sports rules are pushing aerodynamic designs further than ever and radically increasing the efforts of teams to maintain optimum performance.” He noted that the front wing will be completely re-engineered, for example, to compensate for the lack of winglets, which are responsible for producing sufficient down force in critical situations. “Now more than ever, CFD continues to be a major factor influencing overall lap times and an indispensable tool in the development of Formula 1 racecars.”

The author thanks freelance writer Ulrich Feldhaus for contributing portions of the material in this article.
New Check Valves Swing into Action

ANSYS multiphysics simulation and design optimization determine petrochemical check valve flow characteristics overnight — a task that otherwise could take years to complete.

By Christophe Avdjian, Design Development Manager, Engineered Valves, Cameron Inc., Vitrolles, France

For over 60 years, Cameron’s Valves and Measurement group has supplied full port check valves for oil, gas and petrochemical applications. Manufactured under the well-known brand Tom Wheatley™, these robust and reliable valves contain a disk-shaped clapper at the end of a pivoted arm that is forced shut by reverse flow of the medium, such as natural gas, crude oil or refined petroleum products. Through this action, the valve provides instantaneous protection of compressors and other upstream equipment costing hundreds of thousands of dollars. Using such valves helps avoid operational downtime expenses that can reach into the millions of dollars.

In one recent project, Cameron’s engineers used a range of technologies from ANSYS to develop a new line of check valves. Primary design goals were to reduce pressure losses in the valve and to increase the time the valve would stay fully open in low-flow conditions — particularly during system startup, when pipelines are often not operating at full flow for weeks, months or even years.

Reducing Pressure Losses

To reduce pressure losses, engineers used ANSYS CFX fluid flow software to locate areas of maximum turbulence inside the valve body. The ANSYS CFX product was used in conjunction with ANSYS DesignXplorer software for design of experiments (DOEs) to optimize the shape of the clapper disk for greater flow efficiency in fully open and partially closed positions. By efficiently performing many sequential fluid flow analyses, ANSYS DesignXplorer software goes through multiple iterations and quickly converges on a target solution: in this case, the disk shape causing the least amount of pressure loss. In this goal-driven approach, Cameron’s engineers used on-screen slider bars to change the various input parameters and immediately view output parameters displayed as color-coded 3-D response surfaces that guide the way to an optimal design.

In this iterative process, the use of ANSYS DesignModeler technology was critical in preparing the range of geometries for efficient part meshing with each iteration, as well as generating the fluid portion of the model for analysis. The tool readily imported CAD geometry from its native Autodesk® Inventor® format and created a parameterized model, enabling engineers to modify component geometries anywhere in the process simply by changing a few key values. Through this process, the engineering team was able to achieve...
an almost three-fold improvement in pressure loss using the new design.

**Longer Open Times**

Similarly, Cameron’s engineers used ANSYS DesignXplorer and ANSYS DesignModeler technologies in conjunction with ANSYS Mechanical software to optimize the weight of the arm and clapper without overstressing or deforming the parts. This simulation reduced the mass and inertia of the assembly up to 50 percent, resulting in less weight pulling down on the clapper and, therefore, longer open time for the valve during low-flow conditions. Open time was further increased by a multi-physics optimization of the angle of the arm and clapper assembly, performed using the ANSYS DesignXplorer tool in conjunction with ANSYS CFX and ANSYS Mechanical technology in the same set of simulations. The end result was a dramatic improvement in low-flow performance, with the valve remaining open for flow rates nearly 75 percent lower than nominal rates of other comparable check valves.

In addition, engineers used the ANSYS Mechanical product by itself in analyzing stresses and non-linear contact in pressure-containing portions of the valve body assembly as well as shock forces and deflections of the clapper in its emergency “slam shut” mode of operation. Cameron’s Tom Wheatley experience combined with the latest tools from ANSYS has enabled the development of a new generation of check valve that provides a higher level of performance than was previously available. The valves are tentatively scheduled to arrive on the market early in 2009.

**Evaluating Full Range of Characteristics**

Cameron’s engineers are also using ANSYS CFX and ANSYS DesignXplorer tools to compile a reference database of valve performance under a wide range of flow rates and fluid types for each of the 50 different sizes and pressure classes of the new valve. The angular position of the arm and clapper assembly depends on the effects of the inlet fluid, gravity and external forces, so typically up to two weeks are required to perform the many detailed calculations for a limited operational range of one valve size. This can result in delays in getting back to the customer with solutions.

Even with ordinary simulation and modeling processes, determining the performance characteristics for every conceivable feed condition for all 50 valves would be a daunting task, taking years to complete. Cameron’s goal was to develop an approach that leveraged the capabilities of technologies from ANSYS for performing the needed calculations practically overnight. Together, Cameron’s engineers and an ANSYS team in Lyon, France developed a streamlined, automated approach that achieved this goal by linking ANSYS CFX software with the ANSYS DesignXplorer tool in such a way as to analyze virtually thousands of inlet flow-rate and angular position combinations without user intervention, for the entire range of feed conditions on each valve size.

During this automated process, ANSYS DesignXplorer software explores a range of possible angular positions of the arm and disk. Multiple meshes are automatically generated...
using the parametric-driven geometry from the ANSYS DesignModeler tool and then imported into ANSYS CFX software to determine pressure loss in relation to valve input flow rate. ANSYS CFX software also computes the torque at the rotation axis of the arm generated by the fluid acting on the clapper.

These ANSYS CFX outputs are collected by ANSYS DesignXplorer software for every successive simulation representing up to 15 different stages of valve closure positions from fully open to fully closed and input flow rate from minimum to maximum. To verify the torque equilibrium conditions of the arm and disk assembly, engineers created a derived parameter in the ANSYS DesignXplorer tool to permit summing three separate torques acting on the assembly: fluid-induced torque, external forces–generated torque, and torque from the weight of the arm and disk as a function of angle and center of gravity.

A goal-driven optimization was then performed to determine the clapper angle for each inlet flow rate at which total torque equals zero — the point at which the valve is in equilibrium — for all valve feed conditions. From a generated sample of 5,000 points, the Cameron engineering team used ANSYS DesignXplorer technology to iterate through the multitude of solutions and generate trade-off plots as it converged to the ultimate objective of a single curve showing pressure loss as a function of input flow rate for valve equilibrium. This line of points — essentially a flow performance curve for a wide range of feed conditions — was generated for each of the 50 valves, thus providing solutions in hours rather than weeks.
Simulation can provide insight into the complex physical phenomena that occur when severe loads are applied over a short period of time. High-speed impacts, penetrations, explosions, fluid structure interaction and other transient physical phenomena with high stress–strain rates are best solved with an explicit method using programs such as ANSYS LS-DYNA and ANSYS AUTODYN software.

Explicit solvers discretize physical models, often created from CAD geometry, by creating a mesh of elements. The conservation equations for mass momentum and energy are solved numerically using explicit time integration. Combining these equations with material models, initial conditions and boundary conditions that are often nonlinear in nature, engineers and analysts can produce accurate results that correctly model complex physical events.

Running simulations involves making trade-offs among accuracy, ease of problem setup and computing time. ANSYS AUTODYN technology, developed over the past two decades, has consistently focused on ease of use and user productivity.

ANSYS AUTODYN software offers multiple solution methods, such as Lagrange to model structural response, Euler to model gas and fluid flows (including high pressure solid deformations in which metals behave as a liquid), and smooth particle hydrodynamics to model hypervelocity impacts and brittle material fracture and flow. These methods can be combined for various regions of a single problem to reach an

Making an Impact

Modern explicit solutions enable the study of blast or high-impact scenarios.

By Bence Gerber, Regional Manager, Explicit Products, and Tham C. Yang, Software Engineer, ANSYS, Inc.
Truck and car bombs have long been weapons used by terrorists to promote their agendas. These stealthy, low-tech weapons are able to produce devastating and paralyzing effects. Homeland security agencies around the world are now investing millions of dollars on a multilayered approach to counter such attacks. Some of the strategies currently employed include constructing barricades and masonry walls that diffract the blast waves from a car/truck bomb to isolate the blast from the building by providing standoff distance between the exterior of a building and vehicle access; exploiting slopes and high ground in natural terrain when selecting sites for critical facilities; and installing blast-resistant windows to reduce the shards and fragments produced from fractured glass. Derivatives of these strategies are tested with experiments prior to implementation.

Experiments are expensive, risky and often difficult to conduct. While it is best to conduct them at a proving ground miles away from urban development, this can be difficult for nations with limited land area. Furthermore, in such a destructive environment, sensitive pressure sensors and expensive high-speed imaging devices are susceptible to damage and may not survive the explosion.

Computer simulation can provide helpful insight in these cases. As an example, ANSYS AUTODYN software is used to model a truck with a rigid plate bed carrying eight barrels (4,000 kg) of ammonium nitrate-fuel oil (ANFO). The ANFO is modeled in an Euler domain, while the truck and cargo container are represented using Lagrange shell elements. The interaction between the expanding ANFO barrels and the cargo container is defined using fully coupled Euler–Lagrange fluid structure interaction.

The simulation characterizes the blast wave formed by the explosive and the fragments created from the truck and containers. More significant is that ANSYS AUTODYN technology is able to predict the damage caused to a building or other structure by the combined loading of the blast wave and fragments. The simulation illuminates how the expanding detonation products coalesce and fracture the walls of the cargo container, producing a cloud of debris and shrapnel. Just like airborne debris from a tornado, debris and shrapnel from a truck blast result in injury and damage. The injury and devastation from the truck explosion in this sample case is exacerbated by the vehicle’s rigid plate. Through the simulation, one can see that the rigid plate, which supports the eight barrels of ANFO, directs the expanding detonation product outward and upward, resulting in more damage.

The rapid increase in available computing power has enabled the technology of explicit simulation to be used for an expanding number of applications ranging from aerospace to mining, manufacturing to biomedical. The ANSYS AUTODYN product is part of a comprehensive suite of software available through the ANSYS Workbench platform, which continues to grow to provide a complete simulation environment.

Predicting the Initial Stages of an Exploding Truck Bomb

Truck and car bombs have long been weapons used by terrorists to promote their agendas. These stealthy, low-tech weapons are able to produce devastating and paralyzing effects. Homeland security agencies around the world are now investing millions of dollars on a multilayered approach to counter such attacks. Some of the strategies currently employed include constructing barricades and masonry walls that diffract the blast waves from a car/truck bomb to isolate the blast from the building by providing standoff distance between the exterior of a building and vehicle access; exploiting slopes and high ground in natural terrain when selecting sites for critical facilities; and installing blast-resistant windows to reduce the shards and fragments produced from fractured glass. Derivatives of these strategies are tested with experiments prior to implementation.

Experiments are expensive, risky and often difficult to conduct. While it is best to conduct them at a proving ground miles away from urban development, this can be difficult for nations with limited land area. Furthermore, in such a destructive environment, sensitive pressure sensors and expensive high-speed imaging devices are susceptible to damage and may not survive the explosion.

Computer simulation can provide helpful insight in these cases. As an example, ANSYS AUTODYN software is used to model a truck with a rigid plate bed carrying eight barrels (4,000 kg) of ammonium nitrate-fuel oil (ANFO). The ANFO is modeled in an Euler domain, while the truck and cargo container are represented using Lagrange shell elements. The interaction between the expanding ANFO barrels and the cargo container is defined using fully coupled Euler–Lagrange fluid structure interaction.

The simulation characterizes the blast wave formed by the explosive and the fragments created from the truck and containers. More significant is that ANSYS AUTODYN technology is able to predict the damage caused to a building or other structure by the combined loading of the blast wave and fragments. The simulation illuminates how the expanding detonation products coalesce and fracture the walls of the cargo container, producing a cloud of debris and shrapnel. Just like airborne debris from a tornado, debris and shrapnel from a truck blast result in injury and damage. The injury and devastation from the truck explosion in this sample case is exacerbated by the vehicle’s rigid plate. Through the simulation, one can see that the rigid plate, which supports the eight barrels of ANFO, directs the expanding detonation product outward and upward, resulting in more damage.

The rapid increase in available computing power has enabled the technology of explicit simulation to be used for an expanding number of applications ranging from aerospace to mining, manufacturing to biomedical. The ANSYS AUTODYN product is part of a comprehensive suite of software available through the ANSYS Workbench platform, which continues to grow to provide a complete simulation environment.
The head and neck of a soldier represents 12 percent of body area but receives up to 25 percent of all “hits” during combat. For this reason, the use of helmets is critical and as old as warfare itself. Today, military forces are protecting their troops with helmets manufactured from lightweight composite fiber materials such as para-aramid (KEVLAR®, Twaron®), ultra-high molecular weight (UHMW) polyethylene (Spectra®, Dyneema®) and phenylenebenzobisoxazole, or PBO (Zylon®), all of which can provide improved comfort and protection.

The standards for the design and development of the next generation of combat helmets require that ballistic tests be used to ensure adequate protection. Creating multiple prototype helmets and testing them can be expensive and time consuming, especially when exploring multiple material and design variations. Simulation can reduce the cost of development and provide an understanding of the critical physical phenomena that result in failure when the helmet is subjected to various threats.

A 9 millimeter full metal jacket (FMJ) bullet weighs approximately 8 grams and consists of a brass jacket with a lead core. While the kinetic energy from the projectile may not be sufficient to rupture the fibers in a ballistic helmet, it is possible to produce deformation and delamination inside the helmet, resulting in brain trauma to personnel wearing the helmet. By simulating an FMJ bullet traveling at 360 meters per second (m/s), engineers have analyzed the side impact of a bullet on a ballistic helmet, as well as the bullet’s own deformation to provide clues on how to mitigate such threats.

The simulation is carried out using ANSYS AUTODYN software, a computer program capable of computing strains, stresses, velocities and propagation of shock waves as a function of space and time. The ballistic response of anisotropic material is more complex and requires extensive material characterization in comparison to isotropic materials. A composite material model that couples the anisotropic constitutive behavior with nonlinear shock response has been implemented in the ANSYS AUTODYN product in cooperation with Fraunhofer Ernst-Mach-Institut and funded by the European Space Agency (ESA). This material model takes into account anisotropic strength degradation, material anisotropy, melting, vaporization and decomposition, and the coupling of volumetric and deviatoric response.

Using this composite material model, the team demonstrated that the composite helmet can defeat a 9 mm FMJ bullet impacting at 358 m/s from the side. The 9 mm FMJ suffers severe deformation on impact and later collapses, reducing its effectiveness in perforating and penetrating the helmet. Although the projectile does not perforate the helmet, it possesses sufficient momentum and kinetic energy to deform the inside of the helmet. The results from the simulation also reveal that the impact region, both on the outside and the inside of the helmet, is accelerated to a high velocity. This high velocity along the thickness of the helmet results in delamination, indicating failure in the fibers. The deformation and fiber failure, however, are not sufficient to compromise the standoff required for the safety of the soldier using the helmet. A similar approach could be followed to study other armor applications as well.
There's a new energy coming from the people of AMD. It's the power of Fusion. It's how we work with our partners to enable next-generation technologies that change lives. It's where customer needs, dreams, and desires bond with our own passion for engineering. It's what's fueled our past, and what will drive our people and partners in the future. This future is Fusion. And it's only from AMD. To learn more, visit fusion.amd.com.
How did you become focused on engineering simulation, and specifically on multiphysics?

I graduated in 1973 with a civil engineering degree from the University of Padua in Italy. I stayed there to pursue an academic teaching career in construction technologies and mechanics. Research in these fields included work in the fledgling technologies of finite element analysis (FEA), computer-aided engineering (CAE) and intelligent digital prototyping (iDP).

Throughout the 1970s and 1980s, computer power increased dramatically and enabled more complex problems to be handled by engineering analysis codes, such as those available from ANSYS, Inc. A growing number of simulation applications were directed at representing the real-world loads and constraints engineers must always take into account — that is, multiple interacting physics such as structural, fluid dynamics, thermal and electromagnetics.

Capabilities for modeling and analyzing multiphysics problems unleashed a wave of engineering tools that have transformed the way engineers design, develop, retrofit and enhance products in an expanding range of industries. The integration of multiphysics simulation into engineering processes was my vision from the start and remains so today.

You recognized the emerging trends in multiphysics simulation quite early. How did you leverage this technology into a commercial opportunity?

During the 1980s, industrial companies quickly saw the benefits of computer-aided engineering, so in 1984 I founded EnginSoft S.p.A. for technology transfer of these solutions — mostly doing consulting in the Italian market. As a complementary enterprise, in the 1990s we set up ESTECO, a research laboratory for engineering optimization and IT technologies, which these days fall within the field of process integration and design optimization (PIDO).

The concept behind PIDO is to refine designs quickly and effectively by blending simulation into product development processes, rather than performing these studies as a separate function. At ESTECO, we subsequently launched modeFRONTIER™ optimization software for companies to implement PIDO in their particular product development activities. Essentially, the software controls the simulation of a wide range of design variables and quickly converges on a solution that best meets all engineering requirements.

Today EnginSoft has a staff of over 200 and more than 800 clients worldwide. We have been an ANSYS channel partner in Italy since 2003 and have had links to CFX software for computational fluid dynamics (CFD) for over 15 years. The long-standing relationship with ANSYS is one of EnginSoft’s major strengths.

In the early years, was multiphysics less of an issue, since individual engineering simulation disciplines such as FEA and CFD were done separately for the most part?

Yes and no. From the outset, EnginSoft applied an approach aimed at multiple disciplines in what we called CAE solutions for the design chain. We developed ancillary software for transferring models and data from one domain to another — for instance, from finite difference-based CFD to FEA-based structural analysis.

This was cumbersome, but it met customer requirements for the physics to be coupled, even though indirectly. So to some extent, what is known today as multiphysics is the approach we used for multiple disciplines two decades ago. Since then, the approach has widened to encompass a range of multiphysics and multi-scale technologies as well as manufacturing process simulation.

How does technology from ANSYS fit into your current range of activities?

The multiphysics approach of software from ANSYS goes hand in hand with our design process optimization concept for multiple disciplines, with ANSYS structural and mechanical modeling software as well as fluid dynamics and meshing technologies.
performing critical functions in the workflow. We have used this combination with considerable success at many companies, including the auto supplier Mazzucconi and the Piaggio motorcycle and motorbike company. (See accompanying sidebar stories.)

We are currently engaged in two European Union collaborative projects with the aerospace and automotive industries that involve multiphysics and design optimization for all levels of a design process, from component to system level right up to the early production concept phase.

What do you see as the major challenges currently facing the engineering community in making further progress with multiphysics simulation?

As multiphysics and advanced modeling methods become more advanced, simulation-based engineering and science will be indispensable in meeting the technological challenges of the 21st century. The process will not be “simulation as usual” for narrow studies of individual parts and assemblies, but it will be focused on complex, interrelated engineering systems and on analysis results that meet specified standards of precision and reliability. Hence, engineering simulation will develop new methods, technologies, procedures, processes and planning strategies. All these will be key elements for achieving progress in engineering and science. To reap these benefits, however, four significant obstacles must be overcome.

First, we must revolutionize the way we conceive and perform simulation. In this respect, the mass success of computer-based engineering simulation may be its own worst enemy, because the knowledge base, methods and practices that enabled its achievements to date now threaten to stifle its prospects for the future because of organizational inertia and a reluctance to implement new approaches.

Second, we must make significant advances in supporting technologies, including those for structuring the way in which models are built and organized. These technologies have a huge impact on the complexity, solution time and memory capacity required, and, even today, some of the most complex turbulent-flow problems cannot currently be solved on the world’s largest and fastest computers. If progress continues at the rate of Moore’s law, such solutions may not become practical for decades unless effective multiscale modeling technologies are developed to represent the entire range of complexities, from minute individual component details up to broad system-level characteristics.

Third, meaningful advances in simulation-based engineering and sciences will require dramatic changes in education. Interdisciplinary education in computational science and computing technology must be greatly improved. Interdisciplinary programs in computational science must be encouraged, and the traditional boundaries between disciplines in higher education must be dissolved for information to be exchanged smoothly between scientists and engineers collaborating within teams from multiple disciplines.

Fourth, because of the interdisciplinary character and complexity of simulation, we must change the manner in which research is funded. Incremental, short-term research efforts are inadequate and instead should be replaced by long-term programs of high-risk research. Moreover, progress in such research will require the creation of interdisciplinary teams that work together on leading-edge simulation problems.

If applied mathematics and computer science methodologies are focused on computational science at this broad scale in overcoming the above barriers, there is ample evidence that developments in multiphysics and related new disciplines could significantly impact virtually every aspect of human experience.

Where do you see engineering simulation going in the next 10 to 20 years?

From the perspective of EnginSoft, simulation-based optimization will undoubtedly be used for more realistic decision-making in support of engineering design, product manufacturing and field service activities. Tremendous strides are already being made in technologies and approaches for managing the huge amounts of simulation-based data.

Also, among the world’s leading engineering simulation software suppliers, ANSYS, Inc. has the right long-term vision and is making significant investments both in the core disciplines of science and engineering and in the development of algorithms and computational procedures for dynamic multiscale, multiphysics applications.

Do I personally think we will get to a point where science fiction becomes science fact within the next decade or two, where design engineers focus most of their efforts on imagining product variants and product innovations while computers churn away in the background spitting out predictions in real time? I really do think these dreams will become reality in my lifetime.
Mazzucconi Uses Simulation Throughout the Entire Process

In one recent project, EnginSoft used multi-physics technologies in the study of a 1.3 liter diesel engine cylinder head, made by the casting and pre-machining supplier company Mazzucconi for Italian automaker Fiat. In this study, ANSYS Structural software was used in comparing residual stresses due to casting process, pre-machining and heat treatment. This set of simulations represented leading-edge engineering simulation technology that required a wide range of physical transient values (temperature and deformation, for example) to be computed using control-volume meshing and transferred to the ANSYS Structural model. “To my knowledge, this is the first time that the overall project and production process of such a complex cast component was thoroughly simulated by working in a single environment,” noted Luca Pirola, technical manager of Mazzucconi. “It’s amazing how powerful the tool is to analyze the logic of the problem, as well as to optimize the entire process and synthesize and document the results for decision-making.”

Piaggio Boosts Motorcycle Engine Performance by 15 Percent

The Research and Development Center for Piaggio group approached EnginSoft to perform multiphysics optimization studies on one of their motorcycle engines. The goal was to shorten product development time and reduce costs by refining the design with engineering simulation instead of numerous prototype test cycles. The major challenge was in developing an environmentally friendly engine that conformed to stringent emissions standards while maintaining high performance in terms of low fuel consumption, reduced noise and high reliability.

Piaggio engineers first created a 1-D functional model representing the entire engine system, taking into account the full set of structural and fluid dynamics parameters for meeting all of the engine power, torque and energy requirements. Pressure, velocity and temperature output values from the 1-D model served as an input for ANSYS Structural software to calculate the structural behavior of engine materials. The 1-D results were also used as an input for ANSYS CFX software to calculate the complex fluid mechanics and conjugate heat transfer of the engine cooling system.
Geometry of the engine cooling system was imported into ANSYS CFX software to study fluid mechanics and heat transfer of the system.

The modeFRONTIER software managed this multi-variable simulation process providing input to the 1-D code (more than 20 parameters were taken into account), along with the physical and geometrical parameters of the gaskets and cylinder heads for both ANSYS Structural and ANSYS CFX simulation. This design methodology met the project time and cost goals and provided an added benefit of improving engine performance by 15 percent. In this way, ANSYS Multiphysics technology and a collaborative optimization approach helped Piaggio gain a significant advantage in a fiercely competitive global market.
In July 2008, ANSYS, Inc. acquired Ansoft Corporation. As a leading developer of high-performance electronic design automation (EDA) software, Ansoft is world renowned for expertise in electromagnetic, circuit and system simulation. The acquisition of Ansoft is the first foray by ANSYS into the broader electronic design software industry and will enhance the breadth, depth, usability and inter-operability of the expanded ANSYS portfolio of engineering simulation solutions.

Ansoft software allows engineers to simulate component-level behavior, combine this behavior with circuit elements and functional blocks, and optimize system performance under actual operating conditions. To solve the underlying physics of the components, Ansoft tools use finite element and other simulation methods to accurately characterize the electromagnetic behavior. This part of the process is similar to many structural mechanics or fluid dynamics simulations from ANSYS, in that geometry is meshed and solved for a single component. Ansoft also provides tools to allow this model of the component, or a reduced-order equivalent representation, to be used as part of a full system-level simulation.
A History of Innovation

Ansoft has a long history of developing leading simulation solutions for the electronics industry.

By Mark Ravenstahl, Director, Marketing Communications, Ansoft LLC

Ansoft, a leading developer of high-performance electronic design automation software, is now a subsidiary of ANSYS, Inc. We founded Ansoft to meet the design challenges of modern high-performance electronic and electromechanical systems by linking electromagnetic field simulation with circuit and system-level design. Today’s electronics products relentlessly become more dense, operate at higher speeds and grow in functionality. To be competitive in this dynamic market, engineers must be able to simulate the true behavior of these products by including the electromagnetic coupling. Additionally, the convergence of electronics and mechanics in many applications, such as hybrid electric vehicles, has driven the need to unite electromechanical system simulation with rigorous three-dimensional electromagnetic field modeling. Ansoft software allows engineers to simulate component-level behavior, combine this behavior with circuit elements and functional blocks, and optimize system performance under actual operating conditions. The key to Ansoft’s success is solving the physics underlying electrical and electronics products using finite element and other simulation methods and enabling these physics-based solutions to be used in system-level simulations.

The Ansoft and ANSYS combination will address the exploding global demand for more automated and functional products in a wide range of industries: alternative energy, wireless technology, high-speed digital devices, and automotive and aerospace applications. The combination of our two world-class engineering organizations can already deliver many of the tools engineers require to meet these global trends. We are very excited about our future together — working as one company, we will deliver an unprecedented range of simulation technology, from electromagnetics to thermal, fluid flow to structural, physical to behavioral. Together we will deliver Simulation Driven Product Development across the entire spectrum of engineered products.

Zoltan Cendes
Chief Technology Officer and General Manager
Ansoft LLC

Ansoft grew out of research conducted at Carnegie Mellon University by Zoltan J. Cendes, Ph.D., and his colleagues. Dr. Cendes’ early research focused on low-frequency magnetic and electrostatic field computations. The original software developed by Dr. Cendes and his colleagues — Maxwell — was equipped with a powerful Delaunay mesh-generation algorithm that automated the meshing process and made the software very easy to use. In 1984, with the technology developed to the point at which the principals believed it could be turned into a business and Dr. Cendes was convinced that electromagnetics was being underutilized, Ansoft was formed.

In the 1980s, Ansoft started doing cutting-edge research on high-frequency microwave fields. Ansoft developed new types of elements — called edge elements — that ultimately solved the “spurious modes” problem that had been plaguing researchers in finite element modeling of electromagnetic (EM) devices. This development opened the door for the finite element method (FEM) to be employed in electrical engineering applications. In 1990, Ansoft shipped the first version of HFSS (High-Frequency Structure Simulator) technology, which has become the industry standard for computing electromagnetic properties of arbitrary 3-D components and structures. Following that, revenue from HFSS and other Ansoft-developed products for signal integrity analysis and electromechanical system simulation grew at a 25 percent compound average growth rate.

Propelled by the strength of HFSS, Ansoft grew to become a leading developer of high-performance electronic design automation (EDA) software. The unique ability of the products from
Ansoft has expanded its research and development efforts beyond electromagnetics to include circuit and system simulation. Today, the Ansoft product suite focuses on improving physical design by leveraging advanced electromagnetic-field simulators dynamically linked to powerful circuit and system simulation. These capabilities allow engineers to eliminate physical prototypes, maximize product performance and greatly reduce time to market.

With the acquisition of Ansoft by ANSYS, two world-class engineering organizations are brought together — including experienced professionals with depth of knowledge in both simulation and a variety of industries.

Ansoft's target applications are divided into two segments: high-performance electronics and electromechanical systems.
ANSOFT: HISTORY

High-Performance Electronics Products

HFSS is a 3-D full-wave FEA-based product that allows users to extract parasitic parameters (S, Y, Z), visualize 3-D electromagnetic fields (near- and far-field) and generate Full-Wave SPICE models. HFSS utilizes a 3-D full-wave finite element method field solver to compute the electrical behavior of complex components of arbitrary shape and user-defined material properties.

The Nexxim product is an advanced circuit simulator that addresses the increasingly complex, nonlinear and full-wave circuit behavior of gigabit-speed serial interconnects, radio frequency complementary metal oxide semiconductor circuits and GaAs/SiGe radio frequency integrated circuits.

Ansoft Designer software is an integrated schematic and design-management front end linking to Nexxim, HFSS and other field simulators.

The SIwave product analyzes complex PCB and IC packages.

Q3D Extractor software efficiently performs the 3-D and 2-D quasi-static electromagnetic field simulation required for the extraction of resistance, inductance, capacitance and conductance (RLCG) parameters from an interconnect structure. It automatically generates an equivalent SPICE subcircuit model.

The Turbo Package Analyzer (TPA) tool automates the analysis of — and produces lumped or distributed resistance, inductance and capacitance (RLC) models for — all complex semiconductor packages.

Electromechanical Systems

Electromechanical systems are another major segment for Ansoft products. The software is used in the automotive, aerospace and industrial automation industries. The technologies integrate mechanical, electronic and control technology to create synergistic physical systems: This convergence of electronics with mechanics has rendered ineffective iterative design methodologies in which individual design groups are focused on a single aspect of a system. From the initial design stage, modern electromechanical systems are designed with consideration of both the system and the interoperability of components and circuits. The Ansoft electromechanical design solution captures the interactions between electromechanical components, electronic circuits and control logic. This powerful multiple domain approach to design captures the underlying physics that governs all electrical behavior, allowing engineers to accurately model, simulate and validate the component, circuit and system-level performance required for electromechanical system design.

Electromechanical Systems Products

The Maxwell comprehensive electromagnetic field simulation software package assists engineers tasked with designing and analyzing 3-D and 2-D structures, such as motors, actuators, transformers and other electric and electromechanical devices.

The Simplorer multi-domain simulation software tool is used for the design of complex power electronic and drive systems.

RMxprt software speeds the design and optimization process of rotating electric machines.

The PExprt product speeds the design and optimization process of transformers and inductors for power electronics.

For more information on products from Ansoft, visit www.ansoft.com.
The electronics industry faces immense challenges in the global arena in which engineers must address conflicting requirements to increase product functionality while reducing size and weight, lowering energy consumption and complying with stricter government regulations. Pressures from all sides are compounded by shrinking design cycles to meet narrowing windows of business opportunity.

Companies meeting these challenges reap considerable benefits by leveraging growth opportunities in a wide range of electronics segments, including consumer, communications and computational sectors. Furthermore, there is increasing penetration of electronics systems and electromechanical applications in the aerospace, automotive, power generation and power delivery industries. Many of the companies in these industries — large and small, OEMs as well as suppliers — have continued to rely on mechanical and fluid simulation solutions from ANSYS in their product development initiatives. With the recent acquisition of Ansoft, the range of solutions for the electronics engineering community has expanded, complementing the comprehensive capabilities of ANSYS Multiphysics technology.

The resulting breadth of solutions from ANSYS is unparalleled, providing electronics engineers with a range of simulation tools across multiple domains. In assessing reliability, design teams can use ANSYS Mechanical software to study structural and thermomechanical stresses in semiconductor components, electronics packages, printed circuit boards and complete systems. Engineers can incorporate nonlinear phenomena — including solder joint fatigue, delamination and creep — into product design and can conduct modal, shock and vibration analysis. They can also use ANSYS AUTODYN software to conduct drop-test simulation for optimizing product reliability and performance.

ANSYS provides a variety of vertical electronics cooling simulation tools as well as powerful general-purpose computational fluid dynamics solutions to meet the requirements of product miniaturization and high-power densities. ANSYS Icepak thermal management software simulates fluid flow, conduction and radiation heat transfer in various package, board and system-level designs. For evaluating advanced cooling systems, fluid dynamics technology from ANSYS can be used in fan and acoustic design, micro-channel analysis, emersion and phase-change cooling. Fluid dynamics solutions also simulate semiconductor manufacturing processes including etching, photolithography and chemical vapor deposition, as well as semiconductor package manufacturing applications such as encapsulation and curing.

The addition of Ansoft solutions to the ANSYS suite of technologies brings electronics engineers an
expanded range of electromagnetic simulation capabilities. In high-frequency applications, engineers can use the Ansoft flagship HFSS software to conduct full-wave 3-D electromagnetic simulations essential in designing radio frequency and microwave components and systems widely used in radar, antenna, medical device and various wireless applications. Furthermore, HFSS with Ansoft 2.5-D vertical simulation software (Siwave for full-wave and Turbo Package Analyzer for quasistatic analysis) form a powerful toolset useful in design and analysis for signal- and power-integrity applications and electromagnetic compliance. Such analysis is critical in designing high-speed electronics components and systems, such as semiconductor packages, telecommunication equipment, servers, PCs and hand-held devices. These high-frequency electromagnetic capabilities are dynamically linked to Nexxim software, a state-of-the-art time and frequency domain circuit simulator that provides an integration of high-frequency electromagnetic simulation and advanced circuit design and simulation.

For electromechanical and low-frequency applications, Ansoft expands ANSYS product offerings with general and vertical electromagnetic tools as well as system level simulation tools. Maxwell 3-D electromagnetic field simulation software from Ansoft is used for the design and analysis of motors, transformers and other electromechanical devices common to automotive, aerospace and industrial systems. Specialized vertical low-frequency software from Ansoft includes PExprt software for designing transformers and inductors, and the RMxprt tool for analyzing rotating electric machines. These tools are complemented by Simplorer software — a multi-domain system simulation software used for designing high-performance electromechanical systems. These technologies complement the multiphysics, mechanical and fluid dynamics simulation capabilities from ANSYS.

The combined simulation technologies from ANSYS and Ansoft provide the electronics community with extensive solver, meshing, pre- and post-processing, and system- and circuit-level simulation capabilities. ANSYS offers a range of mechanical simulation technologies including automatic contact detection, extensive material models and element types — such as direct coupled-field elements — as well as rigid body and explicit dynamics capabilities. Fluid dynamics technologies include dynamic and moving mesh features, chemical species mixing and reacting flows, specialized models for rotating machinery, and extensive turbulence models. Physics-based meshing from
Signal and power integrity studied with Nexxim and SIwave software from Ansoft

Maxwell software from Ansoft performing 3-D electromagnetic simulation of an electric motor

ANSYS has various algorithms and element types including hexahedral and tetrahedral meshes, and prism, pyramid, quad, tri and bar elements. The acquisition of Ansoft further expands these technologies by adding capabilities such as tangential vector finite element formulations, trans-finite element methods and adaptive finite element meshing capabilities.

To tie all this together and improve efficiency and usability, ANSYS has a variety of advanced infrastructure and data management capabilities that provide for automatic data exchange between various solvers. This data exchange — along with two-way MCAD integration and advanced Six Sigma tools — is achieved through the ANSYS Workbench framework. Management of simulation data and processes is handled by ANSYS Engineering Knowledge Manager (EKM) software, which enables multiple levels of the enterprise to address issues associated with data backup and archiving, traceability and audit trail, process automation, collaboration, capture of engineering expertise and intellectual property protection.

The Ansoft acquisition is a continuation of the ANSYS strategy of providing the engineering community with an integrated and comprehensive multiphysics solution. The benefits of this multi-domain approach are highlighted in the development of hybrid vehicles, for example. Here, engineers not only face the challenge of designing the electric motor and battery systems, but also the need to continually improve aerodynamics, engine performance and stability, underhood thermal management, passenger comfort and crash-test rating. The high end of the automobile industry is seeing more electronics-based features, such as accident avoidance and automatic parking systems as well as navigation devices and entertainment centers.

In such automotive applications, the combined ANSYS and Ansoft technologies provide comprehensive mechanical, fluid dynamics and electromagnetics capabilities for all types of multiphysics applications. Using these tools, engineers can study the inter-related effects of various road or weather conditions on vehicle behavior, for example, as well as the complex interactions among various components. In the end, the multiphysics and multi-system design approach will result in automotive designs that achieve an optimal balance between car performance, gas consumption, weight, environmental impact and safety.

Similarly, in the development of products as diverse as global positioning units, cell phones, MP3 players and other hand-held devices, engineers will be able to use multiphysics analyses to optimize thermal design and mechanical reliability, as well as to conduct extensive electromagnetic analysis and design antenna systems. Most important is that designers in a variety of industries will have a unified design approach at their disposal, which enables simulation to drive the entire design process.
Aligned with the ANSYS Vision

The Ansoft product suite will help deliver benefits to the entire ANSYS engineering simulation community.

By Barry Christenson, Director of Product Management, ANSYS, Inc.

The Ansoft product suite is not merely a strong addition to the considerable tool kit from ANSYS; it will also advance the vision for Simulation Driven Product Development. For the combined community of Ansoft and ANSYS users, some benefits of this depth of solution will be realized immediately; even more value will be revealed in innovative ways throughout long-term product development. The Ansoft technology integration will allow users to perform simulated tests that would otherwise not be possible, a process that is critical to customers exploring and expanding operational boundaries in developing leading-edge products and processes.

Delivering world-class technologies has been part of the ANSYS strategy for developing — and acquiring — new capabilities. Ansoft solver products HFSS and Maxwell, for high-frequency electromagnetic and low-frequency electromechanical simulations respectively, add two world-class leaders in their physics areas.

The ANSYS vision involves a solid base of advanced technologies that enables virtual prototyping. Process compression speeds up the simulation effort. And finally, dynamic collaboration results in innovative products. The ANSYS Workbench platform provides the framework for the process, combining the steps in a truly coupled fashion.

As the Ansoft technologies are fully integrated into the product suite from ANSYS, customers will find that they can simulate their products in ways they never imagined possible.

The ANSYS vision involves a solid base of advanced technologies that enables virtual prototyping. Process compression speeds up the simulation effort. And finally, dynamic collaboration results in innovative products. The ANSYS Workbench platform provides the framework for the process, combining the steps in a truly coupled fashion.
Electric Motors Advanced by “Ultra” Power Storage

Electromechanical simulation tools aid in the design flow of hybrid–electric systems.

By John M. Miller, Vice President Advanced Transportation, Maxwell Technologies, Inc., California, U.S.A.
Marius Rosu, Group Leader Simplorer Modeling, Ansoft LLC

In the energy storage industry, electric double-layer capacitors are becoming widely accepted both in stand-alone applications and in combination with batteries. Also known as ultracapacitors, these devices combine a relatively vast electrode surface area with a molecular-scale charge separation distance, providing capacitances that are several orders of magnitude higher than more common electrostatic or electrolytic capacitors. As the technical and economic benefits of these power-dense components become more widely understood, there is increased interest in the active combination of ultracapacitors as electrical storage elements with advanced batteries, such as nickel metal-hydride and lithium-ion, as the means to offer reliable energy storage over wide temperature and operational limits.

Computer simulation of ultracapacitors and advanced batteries is becoming more widespread as the engineering community becomes better attuned to global climate change and the subsequent demand for more efficient energy storage systems. Examples of ultracapacitors working in combination with advanced batteries continue to proliferate, primarily in the electric and hybrid–electric commercial transportation segments such as transit buses and trains. In such systems, brushless DC (BLDC) motors and their component power electronics play a significant role in increasing the overall system efficiency. These motors supplement the output of the internal combustion engine when extra power is needed. They are also used to start the engine, as opposed to the conventional starter and solenoid method.

When designing a variable-speed drive for a BLDC motor, however, designers face a variety of problems due to the combination of several engineering domains interacting in the device. Some of the major challenges include magnetic design for linear and rotating electrical machines; power electronics design for converters, inverters and DC links; mechanical design for the load profile and oscillations; control design for digital and analog signal components; and multiphysics interaction design for electromagnetic compatibility (EMC) and interference (EMI) requirements.

To address the variable-speed design requirements in a hybrid–electric system, engineers can employ a comprehensive design flow including several electronic design automation tools from Ansoft. Their first step is to select a feasible design for the given rated performance specifications of the electrical machine — for example, current, torque or speed. RMxprt software, a tool designated for the electrical design of rotating machines, allows designers to create a machine model by entering rotor, stator and rating information into a parameterized input module.

To validate the initial design produced by the RMxprt tool, the model is transferred to Maxwell electromagnetic field simulation software to perform a finite element analysis. Maxwell technology provides the designers with critical parameters, including flux linkage versus current for
different angular positions of the rotor. The rotor position information is then used in the controller to synchronize the triggering of each phase of the stator coils with the position of the rotor. To account for unwanted, or parasitic, effects on the bus bar interconnects in the ultracapacitor design, Q3D Extractor parasitic extraction software can be used to compute resistance, inductance, capacitance and conductance (RLCG) parameters and automatically generate an equivalent subcircuit computed at nominal frequency or S-parameters (signal scattering coefficients) calculated for a large spectrum of frequencies.

Ultimately, the analyzed characteristics of different components within the entire power system topology are then assembled within the Simplorer electomechanical simulation software package to verify the complete drive system. When the BLDC model is imported from Maxwell software to the Simplorer tool, the analog-digital characteristics of the circuitry can be modeled. Mixed-signal circuits can then be simulated with both block diagram– and state machine diagram–based representations of the controller, allowing the performance of the system to be optimized.

The simulation of the complete drive system enables the verification of the C-code that will be running on a digital signal processor (DSP) or embedded controller, since such code can be part of the system-level simulation. In Simplorer software, multi-domain components (power electronics, mechanical, hydraulic and thermal) are available to enable more complex study on existing power system design. Engineers can use Simplorer software to create the entire design analysis framework because of the variety of components in the Simplorer tool’s signal characteristics library dedicated to measuring the performance and design quality of the power system.

The motor’s electronic controller contains three-phase bi-directional drivers, which drive high-current DC power and are independently controlled by a block diagram scheme and a state machine diagram. The state machine–based scheme compares the rotor position to determine when the output phase should be advanced. The block diagram uses a hysteresis, or history-dependent, control scheme to chop the phase currents between upper and lower admissible band values in order to allow the electrical motor to develop a sufficient electromagnetic torque to sustain the mechanical load. As an immediate consequence of the hysteresis band control and power inverter switching frequency, the ripples induced in the electromagnetic torque and the harmonic content of the currents affect the overall performance of the system.

At heavy system loads, the ultracapacitor experiences high bursts of power, from both charging and discharging. This eventually will lead to corresponding high carbon loading, which, combined with high current cycling, eventually leads to a reduction in component life. The construction must be robust enough to tolerate high electrical, thermal and mechanical stresses. Hybrid–electric system designers benefit from Ansoft software because the tools provide a comprehensive design flow capable of addressing multi-domain and mixed-signal design by allowing a coupled analysis of the motor, circuit, controller and drive systems.
Microwave Simulation, Macro Benefits

Electromagnetic simulation finds applications in high-performance antenna systems and electronics.

By Lawrence Williams, Director, Business Development
Steve Rousselle, Technical Director, Ansoft LLC

Engineers have long relied on Maxwell’s equations to model the high-frequency performance of wave-guiding structures, such as stripline and microstrip transmission lines, connectors and coaxial lines. Analytical expressions for specific discontinuities, such as the impedance step, open- and short-circuited lines, coupled lines, bends, gaps and junctions, are used by microwave engineers to create matching networks, couplers, power distribution networks, filters and antennas. In the final layout, these models may couple to one another through parasitic electromagnetics, thus creating circuit performance that is different than was intended. Additionally, an engineer may wish to create new components for which there are no available models in the circuit library.

For these and other reasons, electromagnetic field solvers were created that allow the direct solution of Maxwell’s equations to extract accurate models of distributed and parasitic effects. The best electromagnetic simulation tools are capable of full 3-D simulation, allowing engineers to design, analyze and refine microwave components virtually, avoiding costly and time-intensive prototypes and experimental work. Engineers use these tools for Simulation Driven Product Development in order to visualize the electromagnetic fields in their device, understand the device’s electrical behavior to an unprecedented degree, and build virtual products that work as predicted when manufactured.

Today’s microwave and radio frequency (RF) design challenges go far beyond the addition of a few electromagnetics-based models to a circuit. The trend in RF, microwave and high-performance electronics product design is toward accurate prediction of comprehensive system-level behavior with electromagnetic simulation at the core. Engineers now simulate larger and more sophisticated design problems. For phased array antenna systems, for example, designers can simulate the antenna elements as well as the supporting feed network and active circuits behind the array. Other antenna system designers are focusing on the environment in which the antenna operates — the performance of an antenna beneath a radome, for example, or the interaction of a mobile handset and the body of an automobile.

A phased array antenna system, like those on an unmanned aerial vehicle, presents an excellent case study that illustrates the larger system-level designs that are possible today. The ultimate goals of the simulation are to design the antenna and its electronic feed network and then integrate the antenna subsystem into an unmanned aircraft.

Ideally, the ground-mapping X-band antenna system would deliver a flat, equal power versus distance radiation pattern. Achieving the shaped beam is highly dependent upon very precise control of the relative amplitude and phase at each element. This control is often limited by the performance of nonlinear, real-world power amplifiers. To achieve the desired radiation pattern, each antenna in the array receives a different amplitude and phase. As a result, amplifier gain compression will vary among the transistor-based amplifiers in the array. A typical power amplifier will produce a fixed gain and flat phase as a function of input power until a certain point is reached. It cannot produce ever-increasing output power as the input power is increased. Eventually, the device exhibits nonlinear behavior, and its output power compresses while the gain decreases. Using the Nexxim results from harmonic balance circuit simulation of a simple bipolar junction transistor (BJT) power amplifier circuit,
the resulting plot of the circuit gain versus input power shows that the gain is nearly constant until roughly -8 dBm input. Beyond -8 dBm, the gain rolls off quickly.

Transmit power is important to the design of an airborne radar system. Greater power translates into a longer range over which the aircraft can sense aggressors and targets. However, increased power also may result in undesired degradation to the radiation patterns caused by nonlinear behavior of the power amplifiers.

Ideal elevation plane radiation pattern exhibits a shaped-beam region to provide equal power illumination on the ground. Ripples in the main beam are permitted to make the array excitation more realizable. In this case, the pattern’s first four side lobes were suppressed to -30 dB to avoid radiation along and above the horizon. To achieve this pattern, a very specific amplitude and phase distribution along the array was required. Indeed, the amplitude distribution has a dynamic range of over 15 dB. As the input power to the feed is increased, some of the power amplifiers will experience gain compression before others, thus disrupting the prescribed distribution, which in turn degrades the far-field radiation pattern. When the input power increases over 10 dBm, simulation within the Ansoft Designer environment shows that the shaped main beam region is mostly unaffected. However, the first four side lobes begin to rise above the -30 dB level. When the input power reaches 14 dBm, the far-field shows significant degradation due to the nonuniform gain compression across the array.

Powerful 3-D electromagnetic field solvers have long been used by engineers to design complex microwave components. These solvers continue to transform and extend product design, especially when linked with advanced circuit simulation. RF, microwave and high-performance electronics product design now demands accurate prediction of system-level behavior and can include rigorous electromagnetic simulation at the core. Simulations of an active-phased-array antenna system were performed using coupled harmonic balance and finite element simulation.

The effect of the nonlinearities of the transmit power amplifiers was observed to degrade the far-field radiation pattern as the input power was increased. This advanced simulation method allows engineers to fully understand the effects of electromagnetics and nonlinear circuits so that specific design choices can be made.

References
Impeding Interference

Panasonic improves signal integrity design for a remote surveillance camera using electronics design software from Ansoft.

By Hiroshi Higashitani, Panasonic Electronic Devices Co. Ltd., Japan
Aki Nakatani, Ansoft LLC

In a unique network camera device that permits remote visual monitoring for surveillance and security applications, Panasonic used a standard Ethernet connection to transmit video and audio signals, allowing remote monitoring from any location. The camera was designed to rotate, pan and zoom by commands issued by the user. Within the control electronics in the camera body, three module printed circuit boards (PCBs) were connected by a high-speed, low-voltage differential signaling (LVDS) channel with ribbon cables and associated connectors. After building and testing the initial prototype, the designers of the LVDS network camera realized that device performance would be suboptimal, causing them to face a difficult choice: They could either re-spin and test or adopt a new design approach.

Panasonic engineers modeled the full LVDS channel using a combination of HFSS, Nexxim and Ansoft Designer software. The channel included three PCBs (video, mechanical controller and central processing unit) and two Molex® FFP/FPC surface mount connectors. Additionally, the team used the HFSS tool to extract Full-Wave SPICE and S-parameter models for the PCBs. Similarly, they created W-element and 2.5-D planar models for the connectors using Nexxim and Ansoft Designer software. The engineering team then inserted the individual models into a circuit simulator to form the complete channel. With the full channel assembled, the circuit simulator then provided a channel simulator then provided a channel.

Vias are plated holes that connect copper tracks or traces from one layer of a PCB to another.

Pads are surfaces on PCB boards to which components can be mounted.

Traces are the electronic pathways that transmit signals from one component to another.

Timing skew occurs when a clock signal travels along traces and reaches its component destinations at different times.

Today’s printed circuit board (PCB) designers face competing challenges of smaller, higher-density applications coupled with high-frequency and high-speed signaling. A multitude of standards now exist that utilize high-speed serial signaling. The higher speeds give rise to greater demands on PCB designs to meet signal integrity (SI), power integrity (PI) and electromagnetic interference (EMI) specifications. The challenge becomes especially acute for low-cost commercial devices in which traditional signal-integrity design rules may be ignored in exchange for a board with fewer power and ground planes or a higher-density design with less than optimum signal routing.

SI, PI and EMI design once were considered separate disciplines, each with its own design rules, analysis methods and measurement techniques. A more modern approach is to recognize that there is a strong interdependence among the three and that optimum board design requires an integrated approach. A signal-integrity problem, for example, may lead directly to an EMI problem. This article illustrates the new approach, and the important design considerations, through an overview of an LVDS application.
ANSOFT: SIGNAL INTEGRITY

The system’s initial design had a significant impedance problem along the video board. Upon further examination of the video board’s layout, the team found that a pad and via were the root cause of the impedance problems. The impedance mismatch was the result of a step change in the width of the trace located near the via. Beyond the impedance mismatch, the team determined that the original trace routing would also lead to skew.

Panasonic engineers addressed the skew and impedance mismatch in two steps. First, they reconfigured the trace routing so that the total length of each trace in the differential pair was equal; this was accomplished by overlapping the traces. They resolved the impedance mismatch, on the other hand, by eliminating the width step change in the routing to the via and by optimizing the pad and antipad radii. This was done by parameterizing the pad and antipad geometries in HFSS software.

Once they identified optimal routing and via geometries, the engineers focused on the impedance peak of one of the connectors. Polyamide strips were placed on the surface of the connector over certain sections. With their higher permittivity, the polyamide strips caused the local electric fields to be more tightly concentrated. Hence, the capacitance of the transmission line increased, and the characteristic impedance fell. Finally, Panasonic engineers added a common mode noise filter to the circuit to reduce common mode signals while permitting differential signals. Having made these changes, they generated a second impedance map and found that the impedance variations were significantly reduced.

By addressing the signal integrity problems in the PCBs and the connectors, the team confirmed that they had improved the channel’s electromagnetic interference performance. In the initial design, the LVDS signal was scattered whenever it encountered an impedance discontinuity. The scattered energy had to go somewhere; some of it scattered back toward the transmitter, some of it coupled to other propagation modes, especially common mode, and still other energy coupled into parallel plate resonant modes within the PCB. This energy could then radiate to produce unwanted EMI. Solving the SI problem, therefore, had a direct effect on the radiated emissions of the system. Experimental measurements of the camera’s radiated emissions before and after the modifications clearly showed a reduction as well. By adopting a circuit and 3-D electromagnetic cosimulation approach, the design team saved about two months on a second prototype build and about one month on lab measurements.
Decreasing the Shock in Shock Absorbers

Engineering simulation improves valve design for an automotive shock absorber.

By Marcelo Kruger, Geraldo Severi, Martin Kessler and Regis Ataides, ESSS, Florianópolis, Brazil
Sergio Vanucci, Pedro Barau and Robson Iezzo, Magneti Marelli Cofap, Campinas, Brazil

Automotive design requires consideration of a large number of factors including passenger comfort. Suspension design — more specifically, shock absorbers — requires special attention from engineers in order to improve ride quality. Without shock absorbers, a vehicle would have an uncomfortable jolting motion, as energy stored in the spring is released to the vehicle. Shock absorbers dampen spring vibrations by turning the kinetic energy from the springs into heat that is dissipated through a hydraulic fluid.

Engineers at Magneti Marelli Cofap in São Paulo, Brazil — a division of the international company that is committed to the design and production of high-tech systems and components for the automotive sector — have been developing a shock absorber system that is controlled mainly by valves employing circular disks. For low velocities, the fluid passes through small bleeds, or holes, on the disks, with the passage area defined by the number of bleeds. For high-velocity conditions, a different method is used.

In order to improve the shock absorber design and obtain a better understanding of hydraulic fluid flow, engineers from both Magneti Marelli Cofap and Engineering Simulation and Scientific Software (ESSS), an ANSYS channel partner in South America, collaborated on an engineering simulation analysis. The team built a computational model of the shock absorber for the low-velocity condition in order to understand the flow physics that occur in the valve, to evaluate dispersion of forces at the disks and to compare the results against experimental data.

To evaluate the flow pattern and measure the forces, the engineering team used ANSYS ICEM CFD meshing software to generate a computational mesh consisting of tetrahedral and prisms elements and containing 1 million nodes. Simulation of the steady-state flow was performed using the k-epsilon turbulence model in ANSYS CFX fluid flow software. The team considered three configurations of bleeds (two, eight and 16 bleeds) under three velocity conditions for each configuration at the inlet. They obtained the pressure field on the regions of interest at the valve using the CFX-Post post-processor and generated a plot of force (at the valve) versus velocity. The results showed very good agreement with experimental data.

The reliability of the results has given Magneti Marelli Cofap engineers increased confidence in the computational model and, by comparing the different configurations, has provided them with a better understanding of the complexity of the flow pattern. Simulation has imparted useful insights that the engineers now rely upon to make important decisions concerning the improvement of shock absorber efficiency.
During a scheduled outage in 2000, an inspection was performed on a check valve in use at a fossil fuel power plant in the United States. Working with the utility, Structural Integrity Associates found that the cracks circled the valve seat on the inside of the valve body. At that time, they attributed the cracks to thermal fatigue, although creep was always a concern for components, such as this valve, that operate at temperatures of 1,000 degrees F or higher. Structural Integrity worked together with the utility to measure crack depths in order to allow for monitoring of damage progression during future shutdown inspections.

Subsequently, the analysis team agreed to re-inspect the valve body during an upcoming outage scheduled for 2007. For this later shutdown, the inspection included examining the valve for further cracking, incorporating those findings into a simulation designed to predict future deterioration, and, in a short turnaround time, determining if the valve was suitable for continued service with or without repairs.

The fleet of power generation equipment in the United States is of an advanced age, particularly in the case of fossil fuel plants. Many facilities have boilers, piping systems or other components that may be nearing the end of their useful lives due to damage accumulated during operation. Components susceptible to service-related damage typically are inspected to detect this damage during tightly controlled and scheduled shutdown periods, or outages. These outage schedules are often developed years in advance by the utility and have little margin for change.

Damaged components fall into two basic categories: those that are damaged but still viable for continued use and those that are in danger of failure and need to be replaced. Because part replacement is costly, it is most efficient to continue using parts as long as possible. Safety and the desire to avoid component failure that can lead to unplanned shutdowns, however, make it essential that component viability be estimated correctly.

Simulation is used to effectively predict crack growth that could lead to power plant valve failure.

By Dan Peters, Structural Integrity Associates, Inc., Ohio, U.S.A.

The fleet of power generation equipment in the United States is of an advanced age, particularly in the case of fossil fuel plants. Many facilities have boilers, piping systems or other components that may be nearing the end of their useful lives due to damage accumulated during operation. Components susceptible to service-related damage typically are inspected to detect this damage during tightly controlled and scheduled shutdown periods, or outages. These outage schedules are often developed years in advance by the utility and have little margin for change.

Damaged components fall into two basic categories: those that are damaged but still viable for continued use and those that are in danger of failure and need to be replaced. Because part replacement is costly, it is most efficient to continue using parts as long as possible. Safety and the desire to avoid component failure that can lead to unplanned shutdowns, however, make it essential that component viability be estimated correctly.
The first steps involved creating a computer model of the valve using SolidWorks®, which was completed in expectation of the 2007 outage. The developers of the model created and designated its features, based on a drawing of the valve geometry, such that one could modify them in the future to match the actual measurements found during inspection.

Structural Integrity used Creep-FatiguePro® — software developed for the Electric Power Research Institute — to perform a crack growth analysis. As part of this analysis, the Structural Integrity team needed to understand both the pressure stresses experienced by the valve during standard operation and the thermal stresses experienced during the shutdown process.

Due to the desire to keep shutdown time as short as possible, the team set up static and transient stress analyses prior to the actual outage using the ANSYS Workbench framework and the SolidWorks model that had been created. The ANSYS Workbench 11.0 environment was helpful in that it would later provide a seamless and fast interface for updating the model and the analysis in the field based on actual “as cast” dimensions.

The team used the ANSYS Workbench platform to run the complete transient thermal stress analysis coupled with a multi-step static stress analysis. To obtain stresses along a path, the switch to the traditional ANSYS Mechanical interface still had to be made, but it was easily automated, with the file structure in the ANSYS Workbench environment making this process much easier than in prior versions.

The transient thermal stress analysis used a convective condition applied to the internal surface and a step temperature change to model the shutdown event. A multi-step static thermal stress analysis was then run to determine the stresses based on the temperature distribution at various points in time.

The pressure and thermal stresses then were normalized using stress transfer functions and were used as inputs into the crack growth software. The calculation of creep and fatigue crack growth is not a trivial concept. It was handled without difficulty, however, with Creep-Fatigue PRO software and the ANSYS Workbench environment, which was able to export stress analysis results that could easily be used to generate the input to the crack growth software.

Structural Integrity used this methodology, beginning with the measurements taken in 2000, to study the advance of the crack propagation between 2000 and 2007. They compared these results to measurement data acquired in 2007. There was good correlation between the simulation predictions and the measured data, validating the prediction process.

By using this methodology to then analyze the valve for further use, analysis results showed that an expensive replacement of the valve was not immediately needed. The crack growth rates in this valve were low, and the valve could be operated safely potentially for many more years with continued monitoring of the valve at future outages. The crack growth model will be updated with ongoing plant operational data to provide a continuing picture of the crack growth rates for the future. The client was able to make decisions quickly during the outage because the analysis was completed within two days of the completion of the valve inspection, thereby minimizing the length of the outage and avoiding an extremely expensive replacement of the valve.
What’s Shakin’?

The combination of 3-D structural dynamics, ANSYS Workbench and classical rotordynamics modeling techniques helps solve rotating machinery vibration problems.

By Josh Lorenz, Senior Principal Mechanical Engineer, Kato Engineering Inc., Minnesota, U.S.A.

In the electric power generation industry, the ability to design and produce reliable and long-lasting rotating machines is, in part, dependent on the ability to control machinery vibration. Due to the dynamics of rotating machinery parts, mechanical vibration can’t be eliminated completely, but designing machines that meet industry standards for acceptable vibration levels is a crucial part of the product design process. Kato Engineering has been in the business of designing such machines since the 1920s.

Kato Engineering uses 3-D structural modeling tools from ANSYS to increase their ability to simulate and predict machine vibration. Classic rotordynamics modeling techniques involving spreadsheet-style programs with axisymmetric beam element solutions have been around for many years and are very useful tools. However, there are times when the complex 3-D structural details of a rotating machine cannot be cast into the form of an axisymmetric beam element model with sufficient accuracy. For critical applications and new designs in which vibration prediction is of the utmost importance, simulation using software from ANSYS allows Kato Engineering to bridge the gap between rotordynamics spreadsheet modeling techniques and real-world vibration behavior.

Recently, Kato engineers inherited a relatively large common shaft motor–generator design with unfavorable vibration performance. Their task was to modify the design, making it more reliable and easier to produce. This included improving the vibration characteristics of the machine.

In order to tackle this type of problem using structural software tools from ANSYS, the engineers at Kato performed frequency sweep harmonic response analyses using a mixed-element modeling technique. This method combines the efficiency of beam element models — for the rotating portion of the machine — with more structurally complex 3-D shell and solid elements that represent the surrounding stationary structures. These surrounding structures include the machine frame, mounting structure, foundation and other components.

The simulation results provided structural vibration displacements and phase angles at each node of the model for all of the frequencies considered. Using the time-history post-processor, the team at Kato took data from nodes in the model where sensors would be installed during real-world vibration tests of the equipment and examined the predicted magnitude of the vibration response versus frequency. Later, the engineering team compared the simulation results with machine vibration Bode plots obtained during vibration testing of the production equipment.

An important portion of the rotordynamic analyses focused on the bearing connections between rotating and stationary structures. Through the use of ANSYS Parametric Design Language (APDL) and MATRIX27 elements in software from ANSYS, the engineering team allowed for the inclusion of complex bearing stiffness and damping phenomena in the simulation model. They were able to include the bearing stiffness and damping characteristics, as a function of rotational speed as well as all of the cross-coupled stiffness and damping terms that are important in the simulation of the bearing behavior. While these types of bearing behaviors have long been a part of spreadsheet-style rotordynamics programs, the Kato team was now able to efficiently incorporate these behaviors into 3-D full-model vibration simulations using the flexible APDL structure and data input–output options. Speed-dependent bearing stiffness and damping coefficients — calculated using dedicated bearing performance software — were curve-fit and programmed into the structural model using an APDL routine.
The ANSYS Workbench framework has been a very powerful tool for Kato Engineering with regard to simulation model construction time. Using the ANSYS Workbench environment, the team was able to make the transition more quickly from CAD geometry to complex finite element meshes. Where 3-D structural model creation for a machine previously may have taken a week, it was now taking only a day or two with the enhanced geometry and meshing capabilities of ANSYS Workbench.

For this particular application, the Kato engineering team was able to predict the steady-state vibration response of the redesigned machine to a reasonable degree of accuracy using structural simulation tools. While the modeling technique relied on the placement of rotor imbalance forces, which were somewhat nebulous in reality, the team was able to get good correlation between simulation and test results using reasonable assumptions for expected magnitudes and locations of rotor imbalance. By iterating with the simulation model during the design phase, Kato engineers were able to determine that, if they control the imbalance of the rotor to a certain degree and at certain locations, they can expect to meet targets in terms of machine vibrations. The improved steady-state vibration performance of the redesigned machine provided a significant boost for customer confidence in Kato Engineering’s capability to produce a reliable product.
In the petrochemical industry, catalytic cracking is one of the major steps in the process of splitting large hydrocarbon molecules into smaller, more useful components for gasoline and jet fuel. The cracking system itself consists of a reactor and a regenerator that are interconnected by a catalyst pipeline network. During the cracking process, the system undergoes mechanical loading — from wind, internal pressure buildup and the weight of the catalyst material — and experiences thermal stresses caused by the repeated temperature changes to the system’s walls.

At the request of chemical equipment manufacturer JSC Neftehimproekt, the Computational Mechanics Laboratory (CompMechLab) at St. Petersburg State Polytechnical University in Russia performed a 3-D structural analysis of a catalytic cracker, taking into account the effects of external fluid flow as well as overall mechanical and thermal stresses. In their evaluation of how these effects would impact the cracking system, the CompMechLab engineers chose software from ANSYS for the selection of the cracker construction and materials.

A primary goal of the simulation was to choose wall thickness values for the reactor, regenerator and pipeline connections, taking into consideration the physical effects on the system structure at all operating conditions. By extension, this would allow creation of a list of requirements for third-party suppliers of structural components.

Within ANSYS Mechanical software, CompMechLab used multi-layer shell elements, including SHELL131 to perform thermal analysis and SHELL181 for structural analysis. The reactor and regenerator walls consisted of two layers: an external layer of steel and an internal concrete lining. Applying these shell elements also led to a reduced number of degrees of freedom, which saved computational resources.

The weight of the catalyst pipeline network is about half as much as the reactor and regenerator vessels. Bellows expansion joints are used in the construction to compensate for the displacement variations caused by changing temperature loads on the catalyst pipelines and to reduce structural loads on the nose pieces. These joints are deformable parts that independently function in an elastic manner when undergoing axial, lateral or rotational movement. For the global model, the CompMechLab team simulated these components using MASS21 point mass elements in appropriate locations relative to the reactor and regenerator vessels.

CompMechLab’s simulation process focused on analyzing the stiffness of the bellows expansion joints and also the...
forces on the system’s spring bearings, which are used to decrease the gravitational loads acting on the nose pieces. The forces on the spring bearings can be counteracted by varying seven thickness and dimensional parameters. Each of the five bellows expansion joints also has three stiffness values to vary, giving 22 independent parameters in total. For each parameter variation, CompMechLab engineers analyzed two sets of operating conditions: the normal working conditions (temperature range 521 degrees C to 740 degrees C, maximum pressure 0.3 MPa) and the design limit conditions (temperature range 555 degrees C to 790 degrees C, maximum pressure 0.8 MPa).

By varying the 22 parameters, researchers performed a series of computations that focused on decreasing the load on the nose pieces. Included in this process was a computational fluid dynamics (CFD) analysis of the wind’s impact on the system’s external pressure distribution. Using ANSYS CFX software, the analysis team simulated the air flow around the cracker with the built-in shear stress transport (SST) k-ω turbulence model, which is a robust model reliable for a wide class of air flow situations. The wind-induced pressure data from ANSYS CFX output was then interpolated onto the elements of the structural model in ANSYS Mechanical software, thus adding to the load contributions from internal pressure distributions, gravity, thermal stress and forces contributed from the catalyst weight.

The global structural analysis model, which included computing stress distributions on the system and their equivalent displacement vectors, assumed linear behavior of the concrete and steel material. Using codes and standards of the Russian oil and gas industry, CompMechLab engineers determined the maximum allowable stresses for the different parts of the cracking system at particular wall thickness values, taking into consideration safety factors for different operating regimes including startup, normal working conditions and shutdown. Comparing the stresses calculated by ANSYS Mechanical software to those allowed by industry standards, the analysis team was able to validate whether a particular wall material thickness was acceptable or whether the stresses were too high, in which case another analysis iteration would be required using an increased thickness.

Following the global model analysis, CompMechLab created a more detailed submodel for thermostructural and cyclic loading analysis of the upper part of the reactor with consideration of the welded joint between the reactor casing and plenum. The team carried out this step to obtain the reinforcement ring thicknesses in the zones of the nose piece connections to the reactor, which are areas of high stress concentration. Based on these detailed analysis results, engineers selected the zone with the highest stress — known as the critical zone — and performed cyclic strength analysis on that zone.

Following the analysis process using software from ANSYS, the design team was able to select appropriate dimensions for all of the structural component parameters. In summary, the CompMechLab engineers utilized the simulation method developed here for analyses of two different cracking systems and, as a result, were able to perform their analyses and issue their technical report for a greatly pleased client in a period of just six weeks.
Assuming that parts of a dynamic assembly act as purely rigid bodies is like assuming that the earth is flat: The truth won’t be known until the assumptions are challenged. There is always an element of risk involved with challenging the status quo, but, luckily, using ANSYS Flexible Dynamics technology is less risky than falling off the edge of the earth.

When challenged with prototyping a new mechanical assembly, most engineering departments turn to a rigid dynamics software program, and for good reason. The advantages of simulating an assembly as a collection of rigid parts connected by joints are undeniable: It is much faster, more design ideas can be investigated in the same amount of time, and a product development team can be more productive. But this time savings comes at the expense of insight, and, sometimes, what isn’t known about a new design can come back to haunt a well-meaning team. Unknowns can include:

- Will our assembly survive the first cycle, or will one of the parts buckle, break or deform so severely that the system locks up?
- Will the assembly vibrate so much that nobody will buy it?
- Will our warranty department have to deal with the big, expensive problem of material fatigue?
- Is this a huge career-limiting mistake that our design team can’t collectively afford to make?

To gain the insight required to answer the above questions (and many others), part and joint flexibility needs to be included in the simulation.

Rigid dynamics simulation can demonstrate how quickly an assembly’s parts are moving, how fast the parts are accelerating or decelerating, and what the forces are at the joints between the parts at any time during the dynamic transient. The total solution time for many rigid dynamics simulations is often measured in seconds, because the number of degrees of freedom is low and all parts are assumed to be infinitely stiff. This fast solve time makes rigid dynamics extremely attractive to those with looming deadlines.

On the other hand, flexible dynamics provides these same part velocities and acceleration data, plus complete deformation, stress and strain data. While this is the information needed to really understand the design, total solution time is longer. Because of this, relying on flexible dynamics in the early stages of design development has never been commercially viable.

Smart engineers have been trying to combine the benefits of the fast solve times of rigid dynamics with the complete performance information that comes only from running a flexible FEA simulation. Several methods have been developed over the past 20 years with varying degrees of success.

**Rigid Dynamics Loads to Static Simulation Method**

The most basic and most widely used method of combining the benefits of rigid dynamics with those gained by using flexible system modeling is to transfer loads from a rigid dynamics run and use those loads on a structurally static system. This marriage of dissimilar technologies has some pros and cons.
Pro
• Dynamic loading on parts is captured accurately, so there is no need to estimate how far to scale up a static load to approximate a dynamic load. This widely practiced approach is sometimes conservative, and sometimes it is not.
• Static structural simulations are some of the most efficient FEA-based solutions that accurately model flexibility.

Con
• The process forces the engineer to choose the transient time points at which to transfer to the structural static simulation.
• Using this method, it is extremely easy to overlook the worst-case loading combinations for all but the simplest assemblies, so the wise engineer using this method applies a very large margin of safety when relying on results.

Craig–Bampton Method
A more sophisticated technique of combining rigid and flexible benefits is the Craig–Bampton method. Using this technique, the flexibility of a system is captured via a model–dynamic solution. The mode shapes and frequencies, or eigenvalues and eigenvectors, are then fed to the rigid dynamics model so that part flexibility is accounted for during a transient. While less of a forced marriage than the previous technique, the Craig–Bampton method is also blessed with pronounced strengths and weaknesses.

Pro
• A modal analysis is one of the most efficient of all dynamic simulations.
• The rigid dynamics reduced-order model gains flexibility at the lowest computational cost, and this has made the method popular with those requiring additional simulation fidelity.

Con
• The method is inherently limited to linear responses due to its reliance on modal analysis results. This means it is not capable of accurately modeling:
  – Anything other than linear materials: no material plasticity, hyperelasticity or viscoelasticity is possible
  – Real-world nonlinear contact, with or without friction and or changing contact status
  – Large deflection
• The method is complicated and consumes much engineering time. Little has been done to automate, or at least streamline, the linking of the modal results with the rigid reduced-order model, likely because of the inherent limitations of the Craig–Bampton method itself.
• Design iterations are painful. Because there is significant manual interaction and data reading, writing and translating, it is nearly impossible to keep up with changes to a 3-D CAD model.
Financial cost is typically very high because two expensive programs must be used, often from different software companies, and these programs are typically run by two different engineers who have been trained on one system but not both.

Rigid and Flexible Dynamics Method

The most modern method of combining the benefits of rigid and flexible dynamics is to create a general-purpose software system that can be used to model full-rigid dynamics with reduced-order models or a full-flexible dynamics assembly, or any combination thereof. For this method, an engineer uses reduced-order models in pure rigid dynamics and is able to keep pace with rapidly evolving design proposals because of the fast solve times afforded by the explicit solver. To gain further insight, the rigid model is modified with the addition of flexible component(s), and a flexible or rigid and flexible system is analyzed.

While some software suppliers have pieces of the rigid and flexible dynamics method, only ANSYS offers this type of system — and it has been in commercial use for nearly two years. To consummate the relationship between rigid and flexible dynamics, the ANSYS Rigid Dynamics product is used as an add-on to ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics software.

Pro

- A single geometry model is used for both rigid and flexible dynamics. This model is typically an easy-to-visualize 3-D model from ANSYS DesignModeler software or a CAD system.

- The same user interface is employed for both rigid and flexible dynamics, so users of one have very little to learn to be able to run the other.

- Models can be converted from rigid to flexible in minutes in as few as four mouse clicks.

Con

- Design iterations are easy. Change the CAD model, click update, and resolve the rigid, rigid and flexible, or full-flexible model.

- The limitations of the Craig–Bampton method do not apply; that is, you are able to model nonlinear contact as well as material nonlinearities at the same time, if desired.

- While creating a rigid and flexible model with contact and material nonlinearities is easy to do, sometimes these nonlinearities cause conflicting convergence targets for the solver. Overcoming these conflicts and getting a converged solution can require some expertise in nonlinear simulations.

- Solver requirements are higher than either of the previous two methods, which has always been the nature of a full-nonlinear transient dynamic simulation. However, new time integration schemes and parallel processing or high-performance computing can be very effective at reducing CPU demands.

References

Multiphase flow is the simultaneous flow of materials with different states, such as gas, liquid or solid, or with different chemical properties, for example gas bubbles in a liquid or oil droplets dispersed in water. Multiphase flows are commonly encountered in a wide variety of industrial applications, ranging from evaporation in distillation columns to sloshing in fuel tanks, and from spray painting to cyclonic particle separators.

Multiphase flows are much more difficult to model than flows of just a single fluid. A complete description of the flow requires solving mass, momentum and energy equations for each of the phases. These equations are more complex than for single-phase flows because they contain additional terms that govern the exchange of mass, momentum and energy between the phases. Because of the wide range of physical phenomena present and the many possible different flow regimes, the exact form of these interphase exchange terms is not always completely known. Multiphase flow models, therefore, often still include empirically derived terms that continue to evolve as research progresses.

Leading fluid flow simulation products from ANSYS offer a comprehensive suite of multiphase flow models that cover most relevant industrial situations. ANSYS sets the standard for multiphase flow simulation not only by offering the widest range of multiphase capabilities but also through extensive experience and knowledge in the application of these capabilities. Understanding the different types of multiphase flow models will assist in the selection of the most appropriate model for a multiphase application.

Free-Surface Multiphase Flows

One type of multiphase flow is free surface. With free-surface flows, there are two or more immiscible fluids, each of which is described as being continuous in significant parts of the flow domain. There are clearly recognizable regions that contain either one or the other fluid, although the shape and location of these regions may vary with time. These regions are large enough that they can be covered by multipole grid cells in the fluid dynamics model. The shape and location of the interface between the fluids is usually of interest.

ANSYS POLYFLOW software uses a deforming mesh method to calculate the free-surface shape of viscous fluids flowing into an open domain. In this case, as the fluid moves, the domain is remeshed so that the mesh follows the exact shape of the fluid interface. This allows for very efficient and accurate predictions of blow molding and extrusion processes.

ANSYS FLUENT and ANSYS CFX software use the volume-of-fluid (VOF) method to describe free-surface flows.
With the VOF approach, the whole flow domain is meshed with a fixed mesh. The motion and the local volume fraction of the phases are calculated along with the shape of the interface between the phases. At any point in space, there is only one or the other of the two fluid phases, so there is only one velocity field at every location. It is common, therefore, to solve only one velocity field, although in cases in which the velocity difference along the interface is large, robustness and accuracy of the calculations is improved if two separate velocity fields are solved. The shape of the fluid interface does not have to match the shape of the mesh. Different interface tracking methods are available with different levels of accuracy, calculation speed and numerical robustness.

Typical examples of the use of the VOF model are flows such as ships moving through water, dam break scenarios, fuel tank sloshing, stratified flows (distinct layers of different fluids), slug flows (very large gas bubbles moving through a liquid in a pipe) and droplet breakup at inkjet printer nozzles.

**Dispersed Multiphase Flows**

In dispersed multiphase flows, there is one continuous phase and one or more dispersed phases. The dispersed phases consist of many discrete small droplets, bubbles or particles that are distributed throughout the continuous phase. Usually, the size of these is small compared with the flow domain, and often they are smaller than the grid cell size. Frequently, there are too many particles to calculate the motion of each individually. The two most common methods used to model these systems in a manageable fashion are the Eulerian...
Particle trajectories in a cyclone separator are shown as calculated with the discrete particle tracking model. The color indicates the particle diameter. The larger particles (red and green) exit from the bottom as intended. Some of the smaller particles (blue) exit from the top, reducing the separation efficiency. Multiphase technology from ANSYS can be used to optimize the separation efficiency for different particle properties and size distributions.

The Lagrangian particle tracking method (LTM) calculates the trajectories of individual particles, drops or bubbles in the continuous phase. It is also known as the discrete phase model (DPM). In practice, this method is most useful when the particles or droplets occupy a small part of the total volume, usually less than 10 percent, and are heavier than the continuous phase. In cases in which the number of particles is too large to calculate, it is possible to simplify the model just by calculating a statistically significant number of particle streams. The effects of the particles on the flow of the continuous phase can be taken into account and vice versa. Mass transfer effects, such as evaporation and condensation, and chemical reactions, such as combustion, can be included. Examples of applications in which the Lagrangian model is used are sprays of droplets in air, such as paint, or solid particles in air, such as fine powders in asthma medication inhalers.

**Steady State or Time Dependent**

Multiphase flow calculations can be either steady state or time dependent. Steady-state calculations are most suitable when the final solution is independent of the initial conditions and there are distinct inflow boundaries for the individual phases. Other situations are commonly modeled as time dependent. Because of the additional equations and the need to model many flows as time dependent, multiphase flow modeling is computationally intensive. Luckily, fluid dynamics software from ANSYS works efficiently on parallel computing systems so that model turnaround times remain reasonable.

Because of the large number of industrial multiphase applications, ANSYS continues to invest in research and development in this important area. The ongoing development effort will result in continuous improvement of the already comprehensive multiphase flow modeling capabilities from ANSYS.
Viscoelastic materials have an interesting mix of material properties that exhibit viscous behavior (like the gradual deformation of molasses) as well as elasticity (like a rubber band that stretches instantaneously and quickly returns to its original state once a load is removed). The clearest way to visualize the behavior of a material containing both elastic and viscous components is to think of a spring (exerting forces to return to its unstressed state) in series with a dashpot (a damper that resists sudden motion, similar to the pneumatic cylinder that prevents a storm door from slamming shut). With these properties, the stresses of a viscoelastic material gradually relax over time when a constant displacement is applied. Conversely, under a constant applied force, elastic strains continue to accumulate the more it is deformed.

Various materials exhibit viscoelasticity, with deformation depending on load, time and temperature. That is, given enough loading over a period of time, many materials will gradually undergo some level of deformation — and the process may speed up as the material gets hotter. For example, an amorphous solid such as glass may act more like a liquid at elevated temperatures, at which its time-dependent response can be measured in seconds. On the other hand, at room temperature, its stiffness is much greater, so glass may still flow, but the time-dependent response is measured in years or decades.

Viscoelastic behavior is similarly found in other materials such as wood, polymers, human tissue and solid rocket propellants, to name a few. Because of this complex behavior, the use of linear material properties is generally inadequate in accurately determining the final shape of a viscoelastic material, the time taken to arrive at that geometry, and the stresses on the part. In these cases, the material’s viscoelasticity must be taken into account in the simulation.

**Viscoelastic Material Models**

In mechanical solutions from ANSYS, viscoelasticity is implemented through the use of Prony series. The shear and volumetric responses are separated, and the well-known relationships between shear modulus $G$ and bulk modulus $K$ are shown below:

$$G(t) = G_0 \left[ \alpha_1 e^{-t/\tau_1} + \sum \alpha_i e^{-t/\tau_i} \right]$$

$$K(t) = K_0 \left[ \alpha_1 e^{-t/\tau_1} + \sum \alpha_i e^{-t/\tau_i} \right]$$

These equations imply that the shear and bulk moduli are represented by a decaying function of time $t$. Simply stated, the user provides pairs of relative moduli $\alpha_i$ and relaxation time $\tau_i$, which represent the amount of stiffness lost at a given rate.

For simplicity, only the shear term $G$ will be considered for subsequent discussions. Start off at time $t$ equal to 0 with the full stiffness (instantaneous shear moduli). Hence:

$$G(0) = G_0 \left[ \alpha_1 + \sum \alpha_i \right]$$

This implies that the sum of the input relative moduli $\alpha_i$ must be less than or equal to 1.0. Consider the extreme case at infinite time, which gives:

$$G(\infty) = G_0 \alpha_1$$

This means that the infinite modulus $\alpha_\infty$ represents the percentage of remaining stiffness. The user-input relative moduli $\alpha_i$, on the other hand, is the percentage of stiffness that is lost, with $\tau_i$ representing the time constant.

**Example Problem**

Figure 1 shows a rubber bumper being compressed by two rigid bodies in which the right body is fixed and the left body is displaced to compress the rubber part in the middle. The rubber bumper was defined with a neo-Hookean hyperelastic material model in ANSYS Workbench Simulation.
Fluids Simulation for Viscoelastic Materials

Viscoelastic materials experience behavior that may be characterized as both viscous and solid. In addition to tools for addressing deformation (ANSYS Mechanical software), the ANSYS portfolio also contains technology in ANSYS POLYFLOW software that can be used to investigate deformation as well as applications that are more fluids related, such as rubber profile extrusion, blow molding or fiber spinning. Both modeling approaches reveal a great deal about the uncommon behavior of the wide range of viscoelastic materials.

A Commands object was inserted under the rubber part with the following contents:

```
tb,prony,MATID,1,1,shear
tbdata,1,0.5,100
```

For this example, a single Prony pair was defined for the shear behavior. The TB command activates the definition of the Prony pair, and the TBDATA command defines the Prony pair values. In this case, a relative modulus of 0.5 was assumed to have a relaxation time of 100 seconds. This means that at infinite time, half of the shear stiffness will be lost at a decay rate, such that at 100 seconds $0.5e^{-1}$, or 18 percent, of the stiffness is relaxed.

Figure 2 shows the maximum equivalent stress in the rubber part as a function of time. Note that the decay is rapid in the beginning. This is due to the exponential function in the Prony series. The response becomes asymptotic, showing that the maximum stress at time equal to 1,000 seconds decreases to nearly half of the maximum stress at the beginning of the solution, as expected. Note that relaxation starts to occur at the beginning of the solution, and this model experiences a multiaxial state of stress, explaining why the long-term maximum stress value is not exactly half of the peak value.

Figure 3 displays the model at time equal to 10 seconds. Note the self-contact that occurs due to the large imposed displacement. A comparison with Figure 4 — displaying equivalent stress at time equal to 1,000 seconds — shows not only the reduction of stress but also some redistribution that occurs due to stress relaxation.

In this way, the Prony series provides an effective tool in mechanical solutions from ANSYS for calculating deformation of materials where stiffness changes as a function of loading, time and temperature. Contact the author at sheldon.imaka@ansys.com for the complete paper from which this column is excerpted.
Small Bubbles, Large Benefit

Air injection simulations show promise for reducing ship drag by injecting air bubbles to reduce skin friction along the hull of a vessel.

By Takafumi Kawamura and Asako Murakami, Department of Systems Innovation, The University of Tokyo, Japan

Given the recent rise in energy costs and the desire to reduce CO₂ emissions, demand for energy-efficient ships is growing. Fuel consumption for a ship can be reduced by lowering the resistance of the hull as it moves through the water. This resistance can be decomposed into skin friction, form drag and wave drag. Although the relative importance of these three components varies by vessel type, skin friction is generally the largest constituent. For relatively slow-moving large vessels such as tankers, skin friction may represent as much as 70 percent of total drag. Therefore, researchers in the marine hydrodynamics field have been making a concerted effort to reduce skin friction.

The concept of using air bubbles to reduce skin friction is not new. This effect has been confirmed in many bench-scale studies, but there are significant problems to overcome before applying this method to actual vessels. The energy required to pump air to the bottom of the ship increases in proportion to the flow rate and the draft of the vessel. Additionally, the efficiency of the propeller decreases in the bubbly flow regime. In order to achieve a net energy savings, the position of the air injectors, the mean bubble diameter and the volumetric flow rate of air must be optimized.

In a recently conducted full-scale study using a 126-meter-long cement carrier, researchers at the University of Tokyo confirmed that fuel consumption can be reduced by 5 percent by injecting air bubbles into the boundary layer at the bottom of the hull near the bow. In this project, the research team used fluid flow simulation to predict the distribution of air around the hull and estimate the reduction in skin friction. They developed an original bubble flow model for modeling the air volume fraction and the velocity of the air phase and implemented it in ANSYS FLUENT software using a set of user-defined scalar (UDS) transport equations. This bubble model was tuned against data collected from bench-scale experiments.

The ANSYS FLUENT simulations revealed that 34 percent of the wetted surface area of the hull is covered with air bubbles and estimated a 10 percent reduction in total resistance.

The computational results also suggested that energy savings are greater in the ballast condition than in the fully loaded condition. These predictions were consistent with results obtained from tests conducted with the cement carrier vessel.

In the near future, it is possible that many ships will be equipped with air bubble injection systems. For designing such ships, computational flow simulation will be a powerful tool given the costs of full-scale testing and the difficulties associated with scaling up bench test results to actual vessel performance.

References

For many, the romantic dream of winter weather is sitting beside a roaring fire with a hot drink while snow drifts gently to the ground outside. However, the reality can be more of a nightmare. According to the National Snow and Ice Data Center in Colorado, U.S.A., seasonal snow affects up to 33 percent of the earth’s total land surface, 98 percent of which occurs in the Northern Hemisphere. With snow accumulation come transportation challenges, natural events such as avalanches, and increased demands related to maintaining our human support infrastructure, including electricity and fuel supply. To better address these challenges, researchers are taking a closer look at snow deformation — a factor that is related to each of these scenarios.

Snow, which is actually an aggregate of ice grains, can be considered at three scales: the micro-scale (10^{-3} meters), the local scale (1 meter), and the landscape scale (10^4 meters). Landscape-scale snow properties can be derived from local-scale snow data using distributed or statistical models. Local-scale snow properties, however, depend strongly on heterogeneities and layering at the micro-scale. Representative elemental volumes (REVs) are able to model local-scale snow; however, they do not take into account micro-mechanical factors that are fundamental in explaining deformation.

To account for the micro-scale factors, researchers at the Cold Regions Research and Engineering Lab (CRREL) use a discrete element method (DEM) to explicitly model the dynamics of snow particle assemblies and the micro-mechanical interaction processes between the grains. DEM was selected for its ability to model materials that undergo large-scale discontinuous deformations that depend on micro-scale contact processes, internal breakage of contact bonds and compaction of broken fragments.

CRREL’s DEM model, called µSnow, stores the particle shapes, velocities and locations; finds contacts; calculates contact forces and moments; calculates conditions for contact bond formation, growth and rupture; and calculates movement for each particle within the aggregate. This explicit representation provides a way to identify the important processes that control snow deformation and directly compare physical experiments to simulation.

Recently, CRREL began combining a fluid flow model with µSnow to account for the flow and diffusion of air and water vapor in snow. Researchers simulated pressure-driven air flow through 3-D models of snow samples and found that modeled results closely matched measured values.

In the future, this methodology for addressing the complexity of the problem by providing a way to incorporate individual mechanisms into the larger snow model can lead to better predictions and understanding for a whole range of snow-related scenarios. So as you sit having happy thoughts while snow gently falls outside your home, think of the science that lies behind the safety of snow tires, avalanche best practices and making sure that your home stays warm through those winter months.
Technology changes everything

Sometimes the change is minor.
And sometimes it rocks an industry.
Transform your business with ANSYS and HP.

ALTERNATIVE THINKING ABOUT CAE

Partnering with the industry leaders in CAE hardware and software will ensure that you do it all faster, better, and more competitively.

www.hp.com/go/CAE
www.ansys.com

Technology for better business outcomes.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA, U.S.A. 15317

Please send address corrections to AdvantageAddressChange@ansys.com