ANSYS 12.0: Launching a New Era of Smart Engineering Simulation

A full generation ahead of other solutions, ANSYS 12.0 takes product design and development to the next level.

By Jim Cashman, President and CEO, ANSYS, Inc.

The current economic climate has completely changed the way most companies view engineering simulation. Leveraging the power of virtual prototyping to compress the product development process and drive down costs is no longer a choice — it's a requirement for survival in an increasingly competitive environment.

In nearly every industry, driving product development through engineering simulation technology has become a key strategy to develop more innovative products, reduce development and manufacturing costs, and accelerate time to market.

Backed by the unmatched power of ANSYS 12.0 software, progressive companies are taking engineering simulation a step beyond. They have already realized the enormous strategic benefits of virtual prototyping — and are now seeking more from their investments in simulation. ANSYS 12.0 enables these forward-looking companies to maximize the efficiency of their simulation processes, to increase the accuracy of their virtual prototypes, and to capture and reuse their simulation processes and data. This next level of performance signals a new era of Smart Engineering Simulation, in which product innovations can be realized more rapidly, and more cost effectively, than ever before.

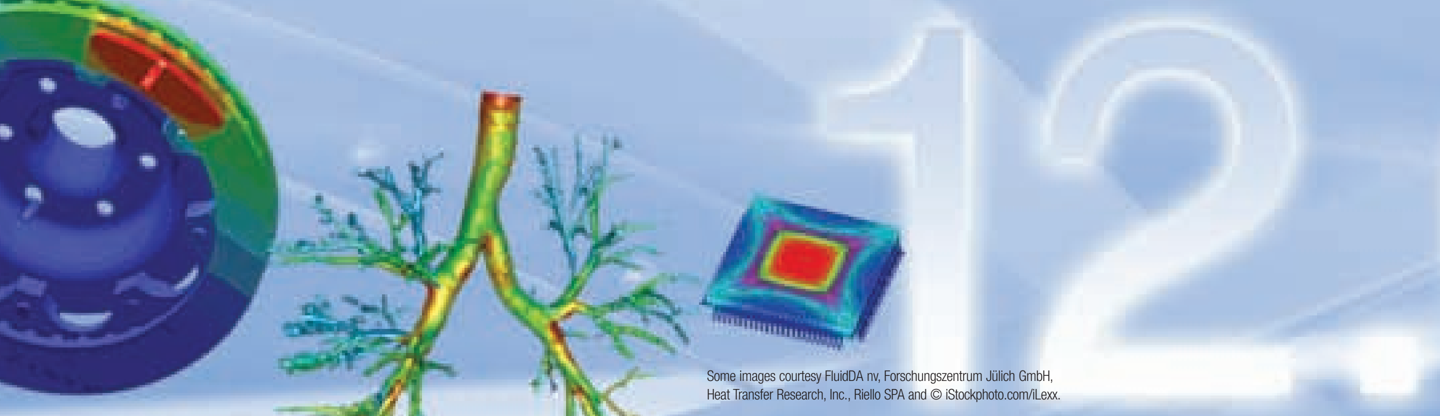
There is no company better qualified to launch this new era. ANSYS has led the engineering simulation industry for nearly 40 years, revolutionizing the field of engineering simulation in much the same way that the internet and desktop publishing have revolutionized the broadband distribution of information. As a direct consequence of a long-standing commitment to simulation, ANSYS is the only company offering advanced simulation technologies that span all key engineering disciplines — and bringing them together in an integrated and flexible software platform designed specifically to support Simulation Driven Product Development.

Over the years ANSYS has made significant technology investments, acquisitions and partnership to ensure continuing leadership. We recognize that every technology breakthrough or market accomplishment has only been a stepping stone to our vision. Reflecting these investments — as well as the acquired wisdom of four decades in this industry — ANSYS 12.0 represents the fullest expression of our leadership position. It is the most comprehensive engineering simulation solution available today.

While the following pages offer a wealth of detail, I'd like to focus on the high-level benefits that our customers will realize as they leverage the full depth and breadth of ANSYS 12.0 to make product development smarter, better, faster and more collaborative than they ever thought possible.

Smart Technologies = Smart Simulation

At ANSYS, we have applied our long history of technology leadership to create the world's smartest solution for engineering simulation — more automated, repeatable,



Some images courtesy FluidDA nv, Forschungszentrum Jülich GmbH, Heat Transfer Research, Inc., Riello SPA and © iStockphoto.com/iLexx.

persistent and intuitive than existing products. The groundbreaking ANSYS Workbench 2.0 platform is a flexible environment that allows engineers to easily set up, visualize and manage their simulations. ANSYS 12.0 offers unequalled technical breadth that allows customers to explore a complete range of dynamic behavior, from frequency response to large overall motion of nonlinear flexible multibody systems. ANSYS has also leveraged its industry-leading capabilities to create an unequalled depth of simulation physics, including the newly integrated ANSYS FLUENT solver, advancements in all key simulation physics, and enabling technologies for meshing, geometry and design optimization. ANSYS Engineering Knowledge Manager allows engineers to easily archive, search, retrieve and report their simulation data via a local machine or a centralized data repository. Not only does ANSYS 12.0 represent the smartest and best individual technologies, but it brings them together in a customized, scalable solution that meets the highly specific needs of every engineering team. Powerful and flexible, ANSYS 12.0 can be configured for advanced or professional users, deployed to a single user or enterprise, and executed on laptops or massively parallel computer clusters. As customer requirements grow and mature, ANSYS 12.0 is engineered to scale up accordingly.

Better Prototypes, Better Products

With its unique multiphysics, high-performance computing and complete system modeling capabilities, ANSYS 12.0 is a complete solution that takes virtual prototyping to a new level of accuracy, realism and efficiency. ANSYS 12.0 captures the response of a completely assembled system and assesses how a range of highly complex, real-world physical phenomena will affect not only individual components but also their interactions with one another. Flaws in product functionality can be recognized before investments are made in full-blown physical prototypes — and ideas that are validated in the virtual world can be fast-tracked to maximize agility and capture emerging market opportunities. Powered by fast and accurate solvers, design optimization with ANSYS 12.0 results in prototypes with a much higher probability of ultimate market success.

Product Design at Warp Speed

ANSYS 12.0 automates many manual and tedious tasks involved in simulation, reducing design and analysis cycles by days or even weeks. An innovative project management system allows custom simulation workflows to be created, captured and automated with drag-and-drop ease. ANSYS 12.0 amplifies the capabilities and outputs of every member of the engineering staff, enabling them to work smarter, to intelligently make design trade-offs and to rapidly converge on the best designs. And, because ANSYS 12.0 is based on the most advanced technology and physics, design and engineering teams can commit to manufacturing operations with confidence — and without investing time and money in exhaustive physical testing.

Redefining Collaboration

Real-world simulation projects often involve a wide variety of engineering personnel — and generate large volumes of data that must be shared across the enterprise. With its broad support of simulation disciplines and native project management system, ANSYS 12.0 allows engineering teams to collaborate more freely, without software barriers or other technology obstacles. Within a single project, several engineers can assess their designs within individual disciplines, as well as easily coordinate multiphysics simulations. The single-project environment reduces redundancies and synchronization errors among different engineering teams. ANSYS Engineering Knowledge Manager also provides the tools to manage the workflow of a group of engineers and a myriad of simulation projects.

At ANSYS, we have always believed that engineering simulation is a sound investment — and today, it is emerging as one of the smartest investments an organization can make. We understand the incredible time and cost pressures under which our customers operate today, and ANSYS 12.0 is specifically designed to help them meet these challenges.

In the new era of Smart Engineering Simulation heralded by ANSYS 12.0, product development teams can work faster and more effectively than ever before — with a greater degree of confidence in their finished products. Because it provides a tremendous opportunity for engineers to design higher-quality, more innovative products that are manufactured faster, and at a lower cost, ANSYS 12.0 makes the most compelling case yet for engineering simulation as a powerful competitive strategy. But we are far from finished: ANSYS 12.0 is a milestone, not the destination, as we continually work to put our tools in the hands of every engineer who can benefit from them. As the power of ANSYS 12.0 is unleashed by imaginative engineering teams around the world, I look forward to the amazing product innovations that will result. ■

Introducing ANSYS Workbench 2.0

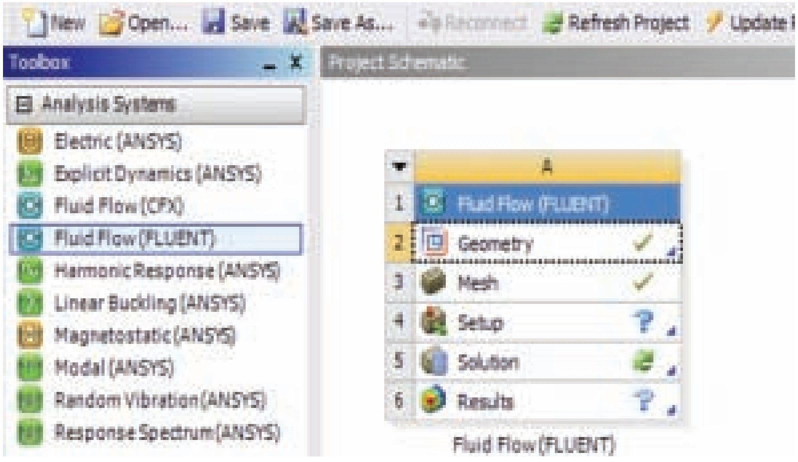
Proven simulation technology is delivered in a truly innovative integration framework.

ANSYS 12.0 delivers innovative, dramatic simulation technology advances in every major physics discipline, along with improvements in computing speed and enhancements to enabling technologies such as geometry handling, meshing and post-processing. These advancements alone represent a major step ahead on the path forward in Simulation Driven Product Development. But ANSYS has reached even further by delivering all this technology in an innovative simulation framework, ANSYS Workbench 2.0.

The ANSYS Workbench environment is the glue that binds the simulation process; this has not changed with version 2.0. In the original ANSYS Workbench, the user interacted with the analysis as a whole using the platform's project page: launching the various applications and tracking the resulting files employed in the process of creating an analysis. Tight integration between the component applications yielded unprecedented ease of use for setup and solution of even complex multiphysics simulations.

In ANSYS 12.0, while the core applications may seem familiar, they are bound together via the innovative project page that introduces the concept of the project schematic. This expands on the project page concept. Rather than offer a simple list of files, the project schematic presents a comprehensive view of the entire analysis project in flowchart form in which explicit data relationships are readily apparent.

Building and interacting with these flowcharts is straightforward. A toolbox contains a selection of systems that form the building blocks of the project. To perform a typical simulation, such as static structural analysis, the user locates the appropriate analysis system in the toolbox and, using drag-and-drop, introduces it into the project schematic. That individual system consists of multiple cells, each of which represents a particular phase or step in the analysis. Working through the system from the top down, the user completes the analysis, starting with a parametric connection to the original CAD geometry and continuing through to post-processing of the analysis result. As each step is completed, progress is shown clearly at the project level. (A green check mark in a cell indicates that an analysis step has been completed.)



The toolbox, at left, contains systems that form a project's building blocks. In this single-physics example, the user drags the system (from left) into the project schematic (at right), then sets up and solves the system, working from the top down through the cells in the system. As shown, the Fluid Flow system (at right) is complete through mesh generation, as shown by green check marks.

Passing files and data from one application to the next is managed entirely by the framework, and data and state dependencies are directly represented. More-complex analyses can be constructed by joining multiple systems. The user simply drags a new system from the toolbox and drops it onto the existing system in the schematic. Connections are created automatically and data is transferred behind the scenes, delivering drag-and-drop multiphysics with unprecedented ease of use.

The ANSYS Workbench environment tracks dependencies among the various types of data in the project. If something changes in an upstream cell, the project schematic shows that downstream cells need to be updated to reflect these changes. A project-level update mechanism allows these changes to be propagated through all dependent cells and downstream systems in batch mode, dramatically reducing the effort required to repeat variations on a previously completed analysis.

Parameters are managed at the project level, where it is possible to change CAD and geometry parameters, material properties and boundary condition values. Multiple parametric cases can be defined in advance and managed as a set of design points, summarized in tabular form on the ANSYS Workbench project page. Design Exploration systems can be connected to these same project-level parameters to drive automated design investigations, such as Design of Experiments, goal-driven optimization or Design for Six Sigma.

In addition to serving as a framework for the integration of existing applications, the ANSYS Workbench 2.0 platform also serves as an application development framework and will ultimately provide project-wide scripting, reporting, a user interface (UI) toolkit and standard data interfaces. These capabilities will emerge over this and subsequent releases. At ANSYS 12.0, Engineering Data and ANSYS DesignXplorer are no longer independent applications: They have been re-engineered using the UI toolkit and integrated within the ANSYS Workbench project window.

Beyond managing individual simulation projects, ANSYS Workbench interfaces with the ANSYS Engineering Knowledge Manager (EKM) product for simulation process and data management. At ANSYS 12.0, ANSYS Workbench includes the single-user configuration of ANSYS EKM, called ANSYS EKM Desktop. (See sidebar.)

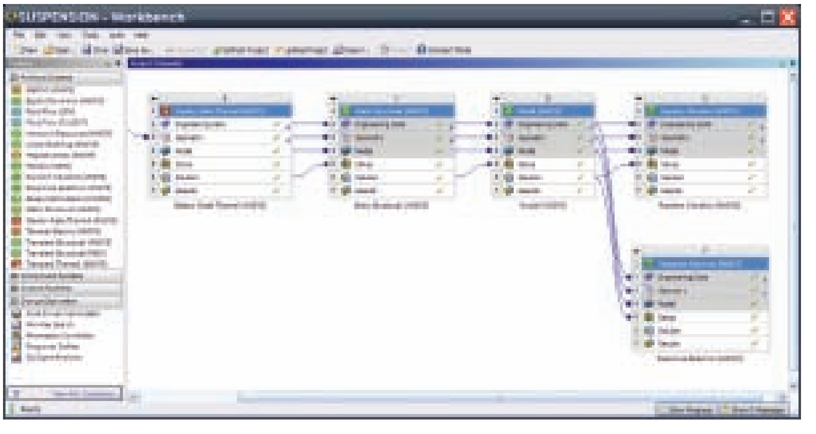
ANSYS Workbench 2.0 represents a sizable step forward in engineering simulation. Within this innovative software framework, analysts can leverage a complete range of proven simulation technology, including common tools for CAD integration, geometry repair and meshing. A novel project schematic concept guides users through complex analyses, illustrating explicit data relationships and capturing the process for automating subsequent analyses. Meanwhile, its parametric and persistent modeling environment in conjunction with integral tools for design optimization and statistical studies enable engineers to arrive at the best design faster. Looking beyond ANSYS 12.0, the ANSYS Workbench platform will be further refined: The aim is to deliver a comprehensive set of simulation technology in an open, adaptive software architecture that allows for pervasive customization and the integration of third-party applications. ■

Judd Kaiser, Shantanu Bhide, Scott Gilmore and Todd McDevitt of ANSYS, Inc. contributed to this article.

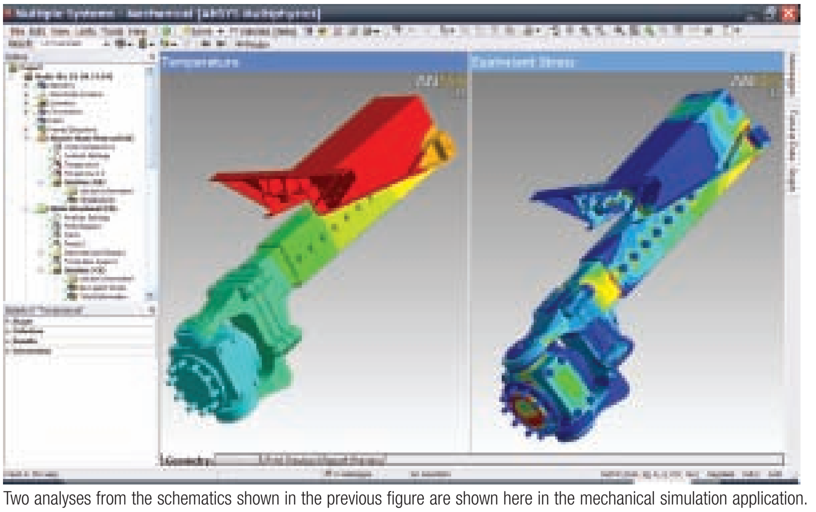
Managing Simulation Data

With the ever-increasing use of simulation, keeping track of the expanding volume of simulation data becomes more and more difficult. The need to be able to quickly locate information for reuse is paramount to increasing productivity and reducing development costs.

ANSYS EKM Desktop is a new tool, integrated in the ANSYS Workbench environment, that facilitates managing simulation data from multiple projects. ANSYS EKM Desktop is a single-user configuration of EKM that allows users to add files from any project to a local virtual repository. Simulation properties and other metadata are automatically extracted (or created) from files when added, and users can tag files with unique identifiers at any time. These attributes can all be used to search and retrieve files based on keywords or complex search criteria. Reports can be easily generated to allow efficient side-by-side comparison of the attributes of related analyses. Search queries and reports can be saved for later re-use. Files that are retrieved can be directly launched in their associated simulation application from within the ANSYS EKM Desktop tool.

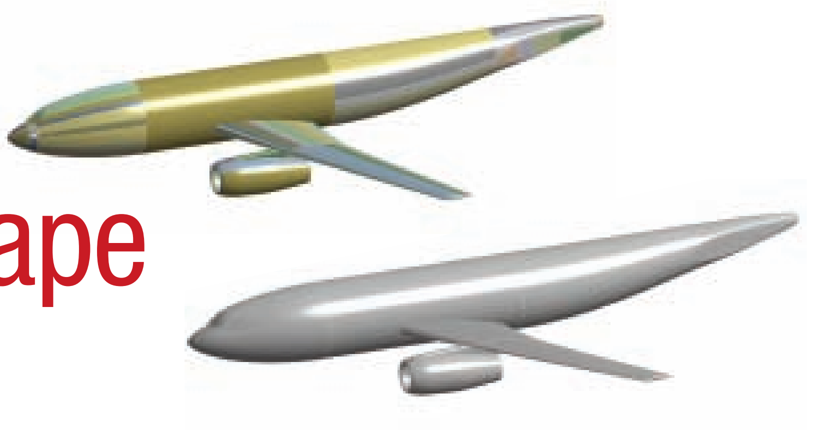


More-complex analyses involving multiple physics can be built up by connecting systems. Data dependencies are indicated clearly as connections. State icons at the right of each cell indicate whether cells are up to date, require user input or need to be updated — for example, whether they are just meshed or fully solved.



Two analyses from the schematics shown in the previous figure are shown here in the mechanical simulation application. Launched from the schematic, individual applications may be familiar to existing users.

Taking Shape in **12.0**



Automated cleanup and repair of imported geometry: New tools automatically detect and fix typical problems, such as small edges, sliver faces, holes, seams and faces with sharp angles. Geometry models can now be prepared for analysis at a much faster pace. These images show an aircraft model before (top) and after (bottom) cleanup.

ANSYS combines depth of simulation with industry experience to provide geometry and meshing tools that realize simulation results faster.

Engineering simulation software users have been known to spend up to 90 percent of their simulation-related time working on pre-processing tasks. By targeting developments in capabilities to increase ease of use, simplifying pre-processing tasks, and increasing the capabilities of pre-processing tools, ANSYS has systematically delivered exciting advances to increase the efficiency of simulation.

ANSYS has combined rich geometry and meshing techniques with its depth of knowledge and experience, and the end result is products capable of harnessing integrated geometry and meshing solutions that share core libraries with other applications. At releases 10.0 and 11.0, ANSYS introduced robust, new meshing capabilities from ANSYS ICEM CFD and ANSYS CFX tools into the ANSYS meshing platform — which provides the foundation for unifying and leveraging meshing technologies, making them interoperable and available in multiple applications. Taking advantage of the enhanced ANSYS Workbench 2.0 framework, the company provides further significant improvements for ANSYS 12.0 geometry and meshing applications.

CAD Connections

ANSYS continues to deliver a leading CAD-neutral CAE integration environment, providing direct, associative and bi-directional interfaces with all major CAD systems, including Unigraphics®, Autodesk® Inventor®, Pro/ENGINEER®, CATIA® V5, PTC CoCreate® Modeling, SolidEdge®, SolidWorks®, and Autodesk® Mechanical Desktop®. Software from ANSYS also supports file-based readers for IGES, STEP, ACIS®, Parasolid®, CATIA® V4 and CATIA V5. At ANSYS 12.0, geometry interfaces have been enhanced to import more information from CAD systems, including new data types such as line bodies for modeling beams, additional attributes such as colors and coordinate systems, and improved support for named selections created within the CAD systems.

For pre-processing larger models, release 12.0 includes support for 64-bit operating systems, and smart and selective updates of CAD parts. The newly introduced ability to selectively update CAD components allows users to update individual parts instead of an entire assembly, thus making geometry updates much faster and more targeted.

Improved surface extension: Users can select and extend multiple groups of surfaces in a single step, a procedure that greatly simplifies the process of closing gaps between parts after mid-surface extraction. The images show a sample model before and after surface extension.

"ANSYS 12.0 will set the stage for major improvements in our design processes. Two of Cummins' core tools, ANSYS FLUENT and ANSYS Mechanical, are coming together in the ANSYS Workbench environment. I am also very pleased to see that geometry import continues to improve, and we have several more meshing options."

— Bob Tickel
Director of Structural and Dynamic Analysis
Cummins, Inc.

Geometry Handling in ANSYS DesignModeler

Geometry modeling in the ANSYS Workbench environment is greatly improved to provide increased automation, greater flexibility and improved ease of use for the task of preparing geometry for analysis. The feature-based, parametric ANSYS DesignModeler tool, which can be used to create parametric geometry from scratch or to prepare an existing CAD geometry for analysis, now includes automated options for simplification, cleanup, repair and defeaturing.

`Merge`, `Connect` and `Project` features have been added for improved surface modeling in ANSYS 12.0. Face and Edge merge operations can be used to easily simplify models by eliminating unnecessary features and boundaries, leading to improved mesh and solution quality. The `Connect` operation can be applied to ensure proper connectivity in models with gaps and overlaps.

Automated cleanup and repair capabilities have been improved in the 12.0 release. New tools automatically detect and fix typical problems, such as small edges, sliver faces, holes, seams and faces with sharp angles. Geometry models can now be prepared for analysis at a much faster pace. As always, analysis settings remain persistent after performing these operations and are updated automatically in response to changes in geometry.

Shell modeling has been enhanced in several ways, including improved surface extensions. The ability to select and extend groups of surfaces greatly simplifies the process of closing gaps between parts after mid-surface extraction. The result is easier modeling of welds, for example.

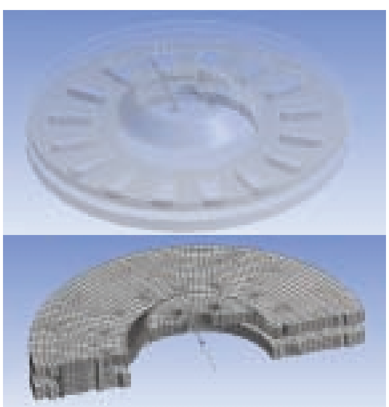
Analysis-specific tools within the ANSYS DesignModeler product now include an automated option to extract flow volumes for fluid dynamics analyses. In addition, several new features, including user-defined offsets, user-defined cross sections and better orientation controls, are available for improved beam modeling for structural analyses.

Improved attribute support is available with ANSYS DesignModeler 12.0. This includes options to create attributes within ANSYS DesignModeler as well as to import additional attributes from external CAD, including named selections, coordinate systems and work points.

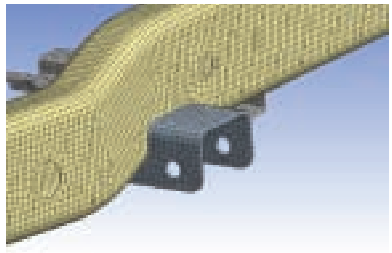
ANSYS Meshing Platform

A primary focus for ANSYS 12.0 has been to provide an automated meshing solution that is best in class for fluid dynamics. With the addition of capabilities from GAMBIT and TGrid meshing applications, major improvements have been made in the automatic generation of CFD-appropriate tetrahedral meshes with minimal user input. Advanced size functions (similar to those found in GAMBIT), prism/tet meshing (from TGrid) and other ANSYS meshing technologies combine to provide improved smoothness, quality, speed, curvature and proximity feature capturing, and boundary layer capturing.

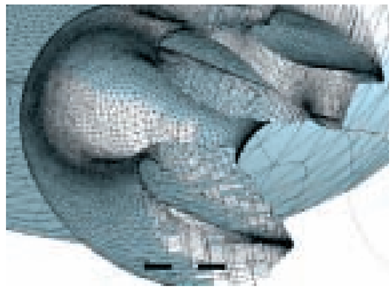
In the area of hex meshing, the traditional sweep and thin sweep methods have seen evolutionary improvements. A new method called MultiZone has been integrated into the ANSYS meshing platform. By combining existing ANSYS ICEM CFD Hexa technology with improvements in automation, MultiZone allows the user to automatically create hex meshes for many complex geometries without requiring geometry decomposition.



MultiZone mesh method: Using the new MultiZone mesh method, a user can mesh complicated models with a pure hex mesh without the need for geometry decomposition. This brake rotor example can be meshed with a pure hex mesh in a single operation.



Thin solid sweep method: Using the thin solid sweep mesh method, complicated sheet metal parts can be easily hex meshed without the need for midsurfacing or welding. The mesh can be generated to conform to the shared interface to increase the accuracy and speed of the solution.

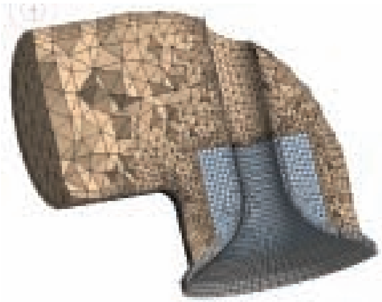


Patch conformal tet method with advanced size functions: With minimal input, ANSYS size function–based triangulation and inflation technology can handle advanced CFD meshing challenges, such as this benchmark aircraft model.

In the area of hybrid meshing, the MultiZone method allows for complicated regions to be meshed with a hybrid mesh (tet, hex-core, hex-dominant), further improving the flexibility and automation of this meshing approach. For more control in key areas of concern, the Sweep and Patch Conforming methods can be employed with conformal inflation layers throughout.

Though many of these enhancements were driven by fluid dynamics needs, they also benefit users of other types of simulation. For example, users performing structural analyses will benefit from the improved automation and mesh quality. Additional meshing enhancements for structural analyses include:

- Physics-based meshing improvements
- Rigid body meshing for contact
- Automated meshing of gaskets
- Improved handling of beams
- Thin solid meshing improvements
- Support for multiple elements through the thickness



Hybrid mesh: Using a combination of sweep and tetrahedral mesh methods, a user can quickly control the mesh in regions of interest to improve the accuracy of the solution without the need for a pure hex mesh (and the time required to generate it).

- Generation of conformal meshes in multi-body parts
- Enhanced and new mesh controls
- Pinch features to help in defeaturing models
- Improved smoothing
- Improved flexibility in size controls and mesh refinement
- Arbitrary mesh matching to improve node linking and solver accuracy

These improvements, though driven by structural analysis needs, provide benefits to the entire spectrum of ANSYS users.

ANSYS ICEM CFD

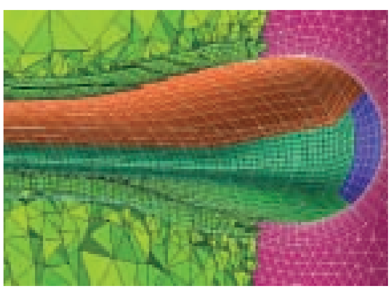
For ANSYS 12.0, ANSYS ICEM CFD meshing development focused on two primary tasks: improved implementation of ANSYS ICEM CFD meshing technology within the ANSYS meshing platform and continued development to enhance the ANSYS ICEM CFD product for interactive meshing customers. Because the ANSYS ICEM CFD integration involves the sharing of core libraries, improvements made for the ANSYS meshing platform also enhance the ANSYS ICEM CFD meshing product (and vice versa).

MultiZone meshing is an example of a crossover technology that has received special attention in both ANSYS meshing and the stand-alone ANSYS ICEM CFD meshing product. This hybrid meshing method combines the strengths of various meshers, such as ANSYS ICEM CFD Hexa and TGrid, in a semi-automatic blocking framework. Within the ANSYS Workbench environment, multizone automation provides multi-source, multi-target and multi-direction sweep capabilities reminiscent of the GAMBIT Cooper tool. In the stand-alone ANSYS ICEM CFD product, this is an excellent way to mesh for external aerodynamics in a semi-automated way that provides rapid hybrid meshing with a high degree of control and quality.

Improvements for ANSYS ICEM CFD 12.0 include process and interface streamlining, new hexa features, BFCart mesher enhancements, mesh editing advancements, output format updates and more. ■

Ben Klinkhammer, Shyam Kishor, Erling Eklund, Simon Pereira and Scott Gilmore of ANSYS, Inc. contributed to this article.

Named selection manager: This new feature allows a user to create and save named selections within CAD systems and then to use them within ANSYS applications. This example uses the named selection manager within Pro/ENGINEER.



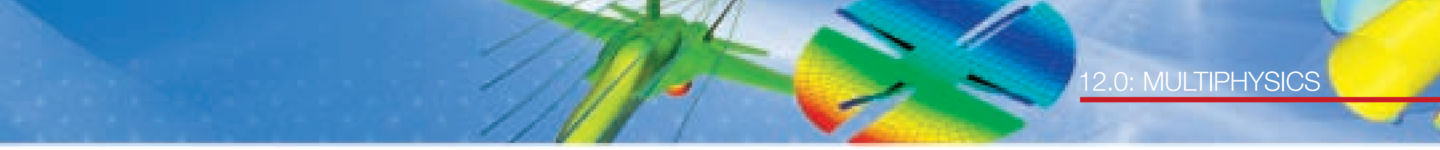
ANSYS ICEM CFD: MultiZone meshing that combines the strength of various meshing tools, automatically generated this hybrid grid for a tidal turbine.

New developments in the ANSYS TurboGrid software are used to create high-quality meshes for bladed components with minimal user input.
Geometry courtesy PCA Engineers.

Enhancements to Turbomachinery Tools

With release 12.0, a number of enhancements have been incorporated into ANSYS BladeModeler, the design tool tailored to bladed geometries for rotating machinery. Within the BladeGen component, the integrated tools for determining initial blade shape and size (which were developed in conjunction with partner PCA Engineers Limited) have been expanded to cover centrifugal compressors and axial fans in addition to radial turbines and centrifugal pumps. The other component of ANSYS BladeModeler, BladeEditor, includes new blade geometry modeling capabilities to create and modify one or more bladed components. As an add-in to ANSYS DesignModeler, ANSYS BladeModeler provides access to ANSYS DesignModeler's extensive functionality to create non-standard geometry components and features.

ANSYS TurboGrid software includes a number of evolutionary improvements in release 12.0, and introduces a completely new meshing technology. This tool fully automates a series of topology and smoothing steps to largely eliminate the need to manually adjust mesh controls, yet still generates high-quality fluid dynamics meshes for bladed turbomachinery components.



Multiphysics for the Real World

In ANSYS 12.0, multiphysics capabilities continue to increase in flexibility, application and ease of use.

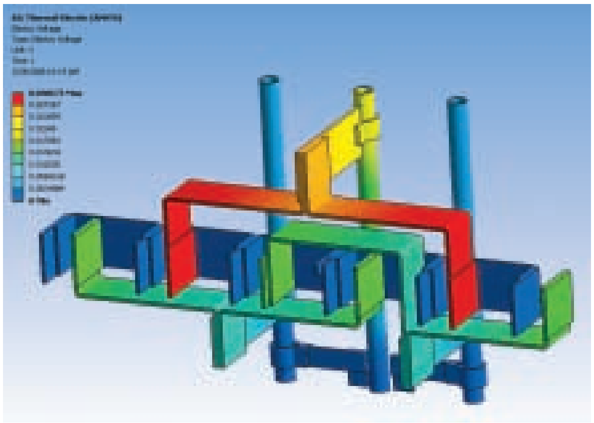
Continuing to build on the foundation of prior releases, ANSYS 12.0 expands the company's industry-leading comprehensive multiphysics solutions. New features and enhancements are available for solving both direct and sequentially coupled multiphysics problems, and the ANSYS Workbench framework makes performing multiphysics simulations even faster than before.

ANSYS Workbench Integration

The integration of the broad array of ANSYS solver technologies has taken a considerable step forward with release 12.0. The ANSYS Workbench environment has been redesigned for an efficient multiphysics workflow by integrating the solver technology into one unified simulation environment. This platform now includes drag-and-drop multiphysics, which allows the user to easily set up and visualize multiphysics analysis, significantly reducing the time necessary to obtain solutions to complex multiphysics problems.

Another new enhancement to the ANSYS Workbench framework is the support for steady-state electric conduction. There is a new analysis system that exposes 3-D solid electric conduction elements (SOLID231 and SOLID232) in the ANSYS Workbench platform. All the benefits of this popular environment — leveraging CAD data, meshing complex geometry and design optimization features — are now available for electric conduction analysis.

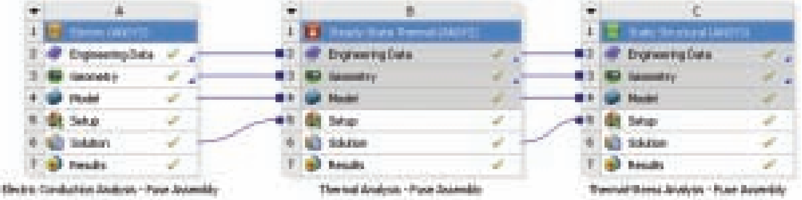
Also new in ANSYS Workbench at version 12.0 is support for direct coupled-field analysis. Relevant elements (SOLID226 and SOLID227) are now natively supported in the ANSYS Workbench platform for thermal–electric coupling. There also is a new analysis system for thermal–electric coupling that supports Joule heating problems with temperature-dependent material properties and advanced thermoelectric effects, including Peltier and Seebeck effects. The applications for this new technology include Joule heating of integrated circuits and electronic traces, busbars, and thermoelectric coolers and generators.



The electric potential for the transformer busbar shown here was analyzed within the ANSYS Workbench environment and required the use of temperature-dependent material properties. Courtesy WEG Electrical Equipment.

Solver Performance

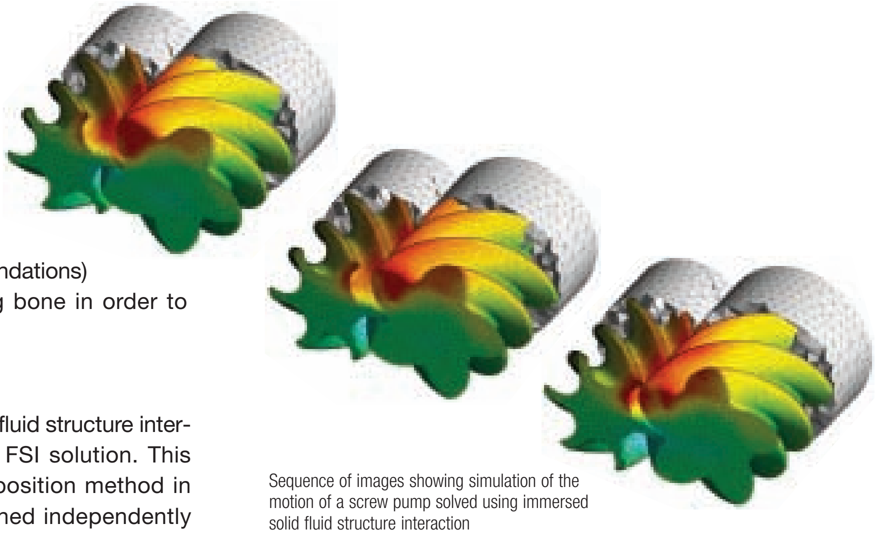
ANSYS 12.0 extends the distributed sparse solver to support unsymmetric and complex matrices for both shared and distributed memory parallel environments. This new solver technology dramatically reduces the time needed to perform certain direct coupled solutions including Peltier and Seebeck effects as well as thermoelasticity. Thermoelasticity, including thermoelastic damping, is an important loss mechanism for many MEMS devices, such as block resonators and silicon ring gyroscopes.



The project schematic shows the multiphysics workflow for a coupled electric conduction, heat transfer and subsequent thermal stress analysis.

Elements

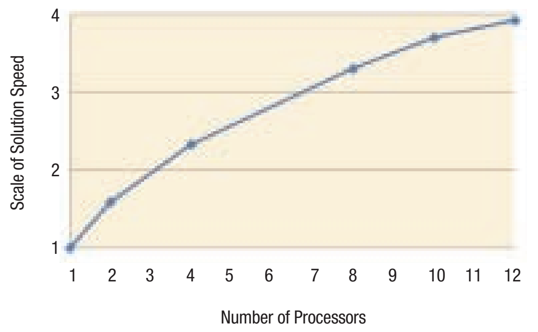
A new family of direct coupled-field elements is available in ANSYS 12.0; these new elements enable the modeling of fluid flow through a porous media. This exciting new capability, comprising coupled pore–pressure mechanical solids, enables multiphysics modeling of new classes of civil and biomedical engineering problems that rely on fluid pore pressures. The elements allow users to model fluid pore pressures in soils (for simulating building foundations) and biometric materials (for modeling bone in order to develop prosthetic implants).



Sequence of images showing simulation of the motion of a screw pump solved using immersed solid fluid structure interaction

Fluid Structure Interaction

One of the major enhancements for fluid structure interaction (FSI) is a new immersed solid FSI solution. This technique is based on a mesh superposition method in which the fluid and the solid are meshed independently from one another. The solution enables engineers to model fluid structure interaction of immersed rigid solids with imposed motion. Rotating, translating and explicit motion of rigid–solid objects can be defined, and the CFD solver accounts for the imposed motion of the solid object in the fluid. This solution technique provides rapid FSI simulations, since there is no need to morph or remesh the fluid mesh based on the solid motion. The model preparation for the new immersed solid technique is also very straightforward: The entire setup for the FSI solution can be performed entirely within ANSYS CFX software. This technology is especially applicable to fluid structure interaction problems with large imposed rigid-body motions, such as closing valves, gear pumps and screw compressors. The method is also useful for rapid first-pass FSI simulations.

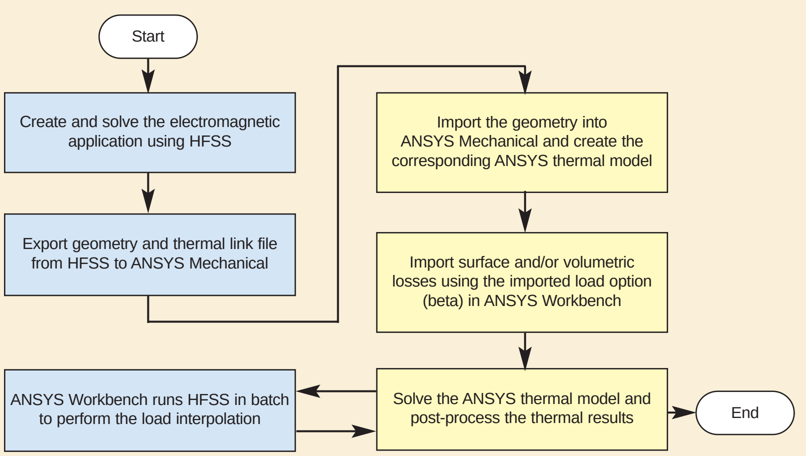


Solution scaling of a thermoelectric cooler model with 500,000 degrees of freedom enables a speedup of four times for 12 processors.

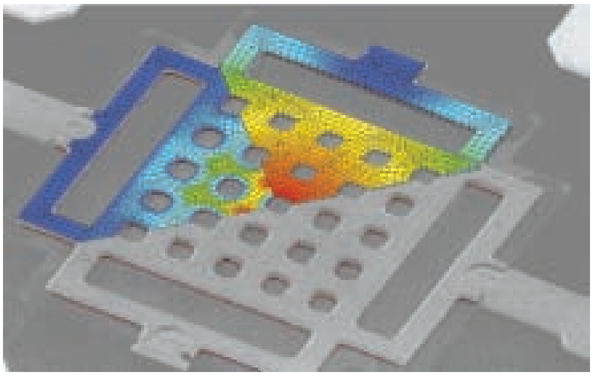
Coupling Electromagnetics

By joining forces with Ansoft, ANSYS can deliver greater multiphysics capabilities — specifically electromagnetics — to the ANSYS suite. The plan to integrate this electromagnetics technology within the existing ANSYS simulation environment started almost immediately after the acquisition. While the combined development team is working toward a seamlessly integrated bidirectional solution, several electromagnetic-centric case studies already have demonstrated the ability to couple electromagnetic, thermal and structural tools within the adaptive architecture of the ANSYS Workbench environment.

For example, a high-power electronic connector used in a radar application to connect a transmitter to an antenna must be engineered from electromagnetic, thermal and structural perspectives to ensure success. The simulation was performed by coupling Ansoft's HFSS software with the ANSYS Workbench environment, using advanced thermal and structural capabilities. Engineers used HFSS to ensure that the device was transmitting in the



Case study procedure of one-way coupling between Ansoft (blue) and ANSYS (yellow) software



The results of an RF MEMS switch solved by coupling the electrostatic, fluid and mechanical behavior of the switch in one analysis using FLUID136 to represent squeeze film effects. Image courtesy EPCOS NL and Philips Applied Technologies.

Another new capability for fluid structure interaction in ANSYS 12.0, FLUID136 now solves the nonlinear Reynolds squeeze film equations for nonlinear transient FSI applications involving thin fluid films. Since the nonlinear fluidic and structural responses are coupled at the finite element level, the solution is very fast and robust for thin fluid film applications. Any squeeze film application can benefit from this technology, including thin film fluid damping often found in RF MEMS switches.

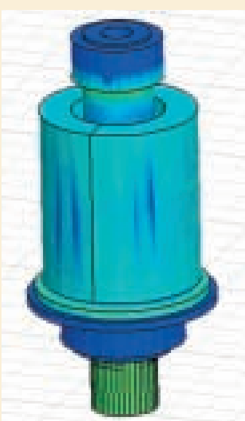
Version 12.0 offers another exciting new FSI capability: the ability to perform one-way fluid structure interaction using ANSYS FLUENT software as the CFD solver. This capability enables one-way load transfer for surface temperatures or surface forces between ANSYS FLUENT and ANSYS mechanical products based on ANSYS CFX-Post. The most appropriate applications include those that require one-way transfer of fluid pressures or temperatures from CFD to a mechanical analysis, such as automotive exhaust manifolds, heat sinks for electronics cooling and turbomachinery.

Multi-Field Solver

The multi-field solver (used for performing implicit sequential coupling) contains a number of new enhancements at release 12.0. The first is a new solution option that controls writing a multiframe restart file. This capability allows a user to restart an analysis from any multi-field time step, which allows for better control over the availability of a restart file with less hard drive usage. Another enhancement is more-flexible results file controls. This capability reduces the results file sizes for the multi-field solver, and it allows for synchronizing the fluid and mechanical results in an FSI solution. The final improvement is new convergence controls for the multi-field solution to provide more flexible solution controls for nonlinear convergence of the multi-field solver. The applications for these enhancements are any multiphysics application using sequential coupling including fluid structure interaction. ■

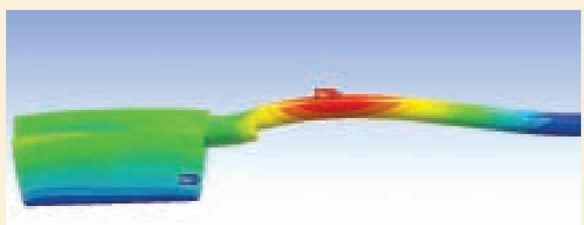
Stephen Scampoli of ANSYS, Inc. and Ansoft LLC technical specialists contributed to this article.

proper path, by calculating the high-frequency electromagnetic fields, power loss density distribution and S-parameters. In such high-power applications, it is critical to determine the temperature distribution to ensure the device stays below temperatures that cause material failure, such as melting. The power loss density results from the HFSS simulation were used as the source for the thermal simulation performed within ANSYS Mechanical software, which simulated the temperature distribution of the device.



Eddy current and conduction loss calculated by Ansoft's Maxwell software

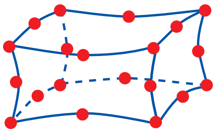
In another case, a valve-actuating solenoid application used a coupled ANSYS and Ansoft simulation to analyze temperature distribution. Maxwell software was used to calculate the power loss from the low-frequency electromagnetic fields within the solenoid. The power loss was used as an input for a thermal simulation performed with ANSYS Mechanical software to determine the temperature profile of the device. Subsequently, the application predicted how the device deformed due to the rise in temperature. Such coupling delivers a powerful analysis framework needed to solve these complex, interrelated physics problems. Thus, engineers can address electro-thermal-stress problems associated with optimizing state-of-the-art radio frequency (RF) and electromechanical components including antennas, actuators, power converters and printed circuit boards (PCBs).



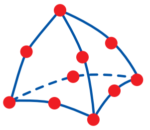
Deformation of the high-power electronic connector can be predicted by combining Ansoft HFSS and ANSYS Mechanical software.

ANSYS Emag 12.0 Generates Solutions

Improved accuracy, speed and platform integration advance the capabilities of low-frequency electromagnetic simulation.



SOLID236
3-D 20-node brick



SOLID237
3-D 10-node tetrahedron

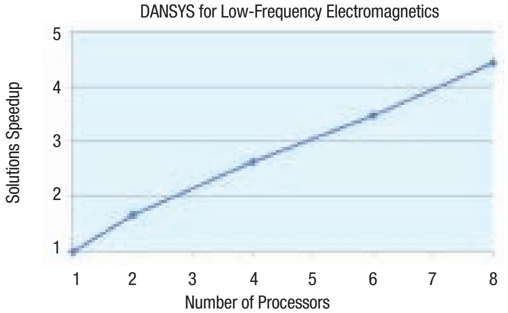
As the combined development teams from Ansoft and ANSYS set out to integrate the world-class Ansoft electronic design products into the ANSYS portfolio, ANSYS customers can benefit immediately from improved and extended electromagnetics capabilities in release 12.0.

Elements

A new family of 3-D solid elements for low-frequency electromagnetic simulation is included in the 12.0 release of ANSYS Emag software. Solid elements (SOLID236 and SOLID237) are available for modeling magnetostatic, quasi-static time harmonic, and quasi-static time-transient magnetic fields. These two elements are formulated using an edge-based magnetic vector potential formulation, which allows for improved accuracy for low-frequency electromagnetic simulation. The elements also provide a true volt degree of freedom — as opposed to a time-integrated electric potential — enabling circuit coupling with discrete circuit elements and simplifying pre- and post-processing for electromagnetic simulation. SOLID236 and SOLID237 also include much faster gauging than prior releases, which significantly reduces overall solution times. Users can apply this new element technology to most low-frequency electromagnetic applications, such as electric motors, solenoids, electromagnets and generators.

Solvers

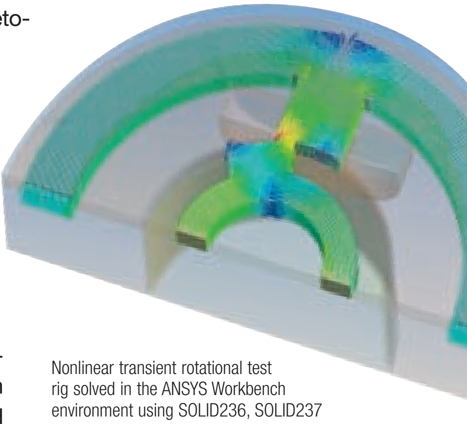
At release 12.0, the distributed sparse solver includes support for low-frequency electromagnetics. SOLID236 and SOLID237 elements support both distributed and shared-memory parallel processing for low-frequency electromagnetic solutions. As a result of faster simulation speeds, users can solve much larger and more complex low-frequency electromagnetic models.



Solution scaling of a SOLID237 model with 550,000 degrees of freedom

ANSYS Workbench Integration

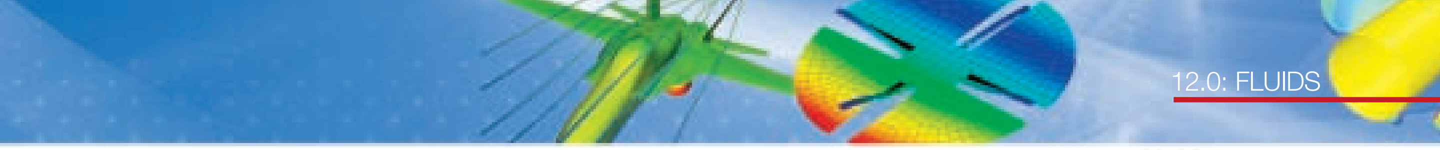
Release 12.0 offers several ANSYS Workbench enhancements for electromagnetic simulation. A new capability facilitates multiple load step analysis for magneto-statics. This allows users to compute the magnetostatic response to time-dependent loading, specifying voltage and current loads with time-dependent tabular data. The results are more flexibility for magneto-static problems with time-dependent loads along with transient simulation for electromagnetics, with the addition of a simple command snippet, within the ANSYS Workbench environment.



Nonlinear transient rotational test rig solved in the ANSYS Workbench environment using SOLID236, SOLID237 and the new stranded conductor option (TEAM24 benchmark)

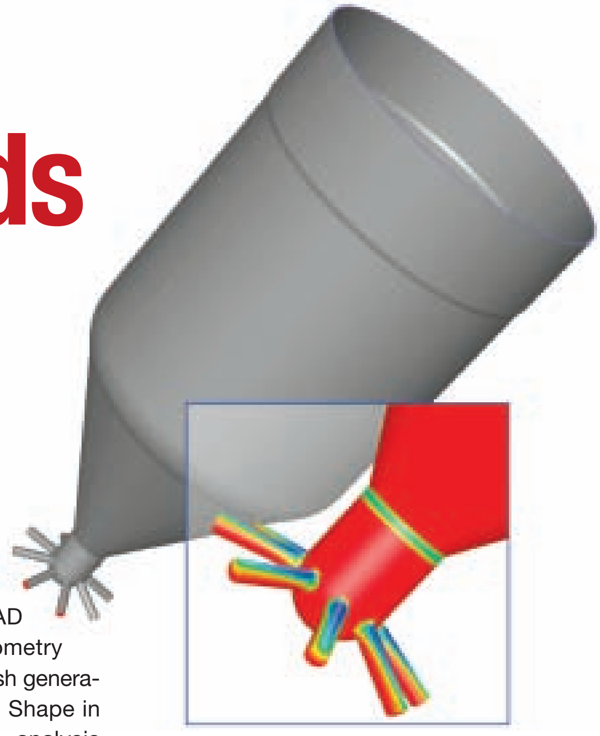
The integrated platform also includes an option for a meshed representation of a stranded conductor. The current density for the new stranded conductor supports tabular loading for the new multi-step magnetostatic analysis. This capability allows for a more accurate representation of current, improves overall simulation accuracy and leverages existing CAD data for coil geometry. This new ANSYS Workbench technology can be applied to any electromagnetic application subject to time-dependent loading, including electric machines, solenoids and generators. ■

Stephen Scampoli of ANSYS, Inc. contributied to this article.



A Flood of **Fluids Developments**

A new integrated environment and technology enhancements make fluids simulation faster, more intuitive and more accurate.



Fuel injector model with close-up of vapor volume fraction contours at the injector surface

With release 12.0, ANSYS continues to deliver on its commitment to develop the world's most advanced fluid dynamics technology and make it easier and more efficient to use. Through its use, engineers can develop the most competitive products and manufacturing processes possible. In addition to delivering numerous new advancements in physics, numerics and performance, ANSYS has combined the functionality of both ANSYS CFX and ANSYS FLUENT into the ANSYS Workbench platform. Customers can use this integrated environment to leverage simulation technology, including superior CAD connectivity, geometry creation and repair, and advanced meshing, all engineered to improve simulation efficiency and compress the overall design and analysis cycle.

Integration into ANSYS Workbench

ANSYS 12.0 introduces the full integration of its fluids products into ANSYS Workbench together with the capability to manage simulation workflows within the environment. This allows users — whether they employ ANSYS CFX or ANSYS FLUENT software (or both) — to create, connect and re-use systems; perform automated parametric analyses; and seamlessly manage simulations using multiple physics all within one environment.

The integration of the core CFD products into the ANSYS Workbench environment also provides users with access to bidirectional CAD connections, powerful geometry modeling and advanced mesh generation. (See the article Taking Shape in 12.0.) Users can examine analysis results in full detail using CFD-Post, also available within the ANSYS Workbench environment.

Multiphysics

In some cases, fluid simulations must consider physics beyond basic fluid flow. Both ANSYS CFX and ANSYS FLUENT technologies provide many multiphysics simulation options and approaches, including coupling to ANSYS Mechanical software to analyze fluid structure interaction (FSI) within the ANSYS Workbench environment.

Another new capability is the immersed solid technique in ANSYS CFX 12.0 that allows users to include the effects of large solid motion in their analyses. (See the article Multiphysics for the Real World.)

General Solver Improvements

ANSYS continues to make progress on basic core solver speed, a benefit to all users for all types of applications, steady or transient. A suite of cases that span the range of industrial applications has consistently shown increases in solver speed of 10 to 20 percent, or even more, for both ANSYS CFX and ANSYS FLUENT software. Beyond core solver efficiency, improvements to various aspects of parallel efficiency address the continued growth and needs of high-performance computing. (See the article The Need for Speed.)

The perennial goal of improving accuracy without sacrificing robustness motivated numerous developments, including new discretization options such as the bounded second-order option in ANSYS FLUENT and the iteratively-bounded high-resolution discretization scheme in ANSYS CFX. Being able to consistently use higher-order discretization schemes means that users will see further increases in the accuracy of flow simulations without penalties in robustness.

User Interface

Ease of use has been enhanced in various ways. Most noticeably, the ANSYS FLUENT user interface has taken a significant step forward by adopting a single-window interface paradigm, consistent with other applications integrated in ANSYS Workbench. A new navigation pane and icon bar and new task pages and tools for graphics window management all reflect a more modern and intuitive interface while providing access to the previous version's menu bar and text user interface.

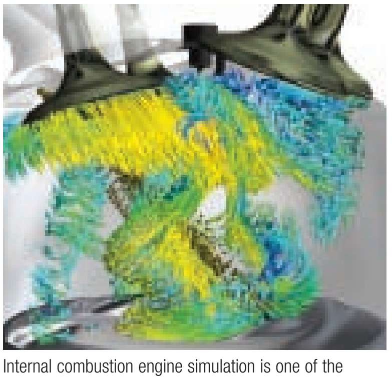
For ANSYS CFX software, a host of improvements have been added to the graphical user interface (GUI). There is a completely new capability that allows users to customize GUI appearance, including the option to create additional input panels. These custom panels provide the ability to encapsulate best practices and common processes by giving the user control over GUI layout and required input.

Specific Focus Areas

Internal Combustion Engines

Internal combustion (IC) engines are a primary target application for the development of numerous features. While this development is driven by the specific needs of IC engine simulations, it benefits many other applications and users:

- New options and flexibility for handling variations in physics complexity required at different phases of analyses
- Further-integrated options and controls for remeshing, including an IC-specific option for setting up an entire engine simulation
- Extensions and improvements to discrete particle-tracking capabilities
- Numerous enhancements to combustion models and their usability



Internal combustion engine simulation is one of the focus applications for ANSYS 12.0. This snapshot from a transient simulation of the complete engine cycle shows the flow just after the intake valves open and the direct injection of fuel. New flow feature extraction options in CFD-Post are used to highlight vortex structures with velocity vectors. Image courtesy BMW Group.

Evolution of the free surface of oil in a reciprocating compressor. The blue area is the gas/oil rotating domain inside the shaft, and the gray surface at the bottom shows the oil level of the reservoir. As the shaft rotates, oil is pumped up due to body forces. Image courtesy Embraco.

Multiphase

Multiphase flow modeling continues to receive a great deal of development attention, in terms of numerics and robustness improvements as well as extended modeling capabilities. ANSYS FLUENT software extends the single-phase coupling technology, introduced previously for the pressure-based solver, to include Eulerian multiphase simulations. This enhancement provides more robust convergence, especially for steady-state flows. ANSYS CFX users will find that improvements to the option to include solution of the volume fraction equations as part of the coupled set of equations make it more broadly usable in applications with separate velocity fields for each phase. Other modeling enhancements include the implementation of a wall boiling model and additional non-drag forces in ANSYS CFX as well as more robust cavitation and immiscible fluid models in ANSYS FLUENT.

Turbomachinery

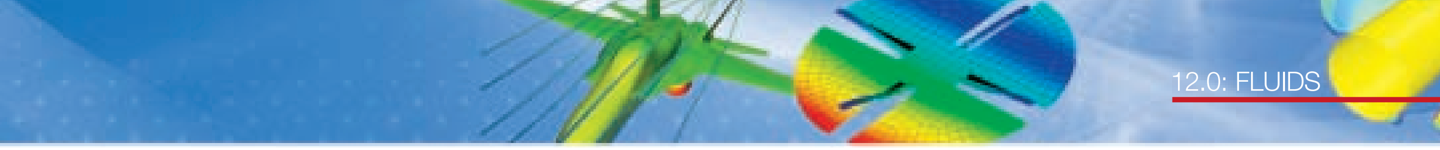
The significant proportion of customers using products from ANSYS for the design and optimization of rotating machinery ensured that this field received a substantial development focus. This latest release contains a variety of enhancements to core solver technology that couple rotating and stationary components more robustly, more accurately and more efficiently. ANSYS BladeModeler and ANSYS TurboGrid, specialized products for bladed geometry design and mesh generation, continue to evolve and improve. (See the Geometry and Meshing article for more details.)

An exciting new development for turbomachinery analysts is the introduction of the through-flow code ANSYS Vista™ TF. Developed together with partner PCA Engineers Limited, Vista TF complements full 3-D fluid dynamics analysis to provide basic performance predictions on one or more bladed components in a matter of seconds, allowing users to quickly and easily screen initial designs.

And More …

These enhancements represent just the tip of the iceberg in new and improved models and capabilities within core fluids products from ANSYS. Some other new developments include:

- Turbulence modeling extensions and improvements
  - Reynolds-averaged Navier–Stokes (RANS) models
  - Laminar–turbulent transition
  - Large eddy simulation (LES)
  - Detached eddy simulation (DES)
  - Scale-adaptive simulation (SAS)
- Ability to use real gas properties with the pressure-based solver in ANSYS FLUENT and, therefore, include these in reaction modeling
- Faster, more accurate chemistry across the board
- Dramatic speedups in view factor calculations in ANSYS FLUENT



"ANSYS CFX 12.0 showed a 30 percent solver speedup in comparison with the previous release. This significant improvement allows us to examine more design variations in the same time, enabling further design optimization and considerably reducing the total development time. This helps Embraco bring our products to the market more quickly."

— Celso Kenzo Takemori
Product and Process Technology Management
Embraco

- Inclusion of convective terms in solids to model conjugate heat transfer in moving solids in ANSYS CFX
- Ability to model thin surfaces in ANSYS CFX
- Much more in areas such as particle tracking, fuel cells, acoustics, material properties and population balance methods

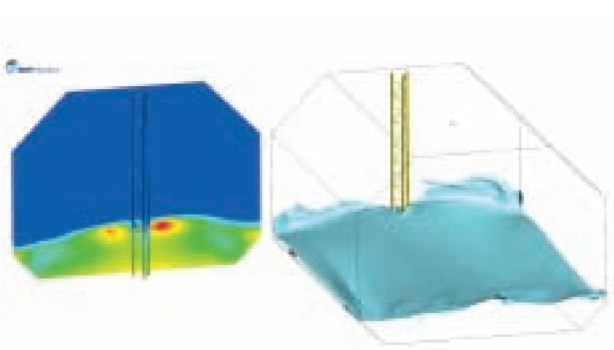
CFD-Post

An exciting introduction is the common post-processing application CFD-Post. The result of combining technologies from both ANSYS FLUENT and ANSYS CFX tools and building upon the well-established CFX-Post application, CFD-Post provides a complete range of graphical post-processing options to allow users to visualize and assess the flow predictions they have made and to create insightful 2-D and 3-D images and animations. The application includes powerful tools for quantitative analysis, such as a complete range of options for calculating weighted averages and automatic report-generation capabilities. All steps can be scripted, allowing for fully automated post-processing. Among the specific enhancements in release 12.0 are the ability to open and compare multiple cases in the same CFD-Post session and the addition of tools to locate vortex cores in the predicted flow field.

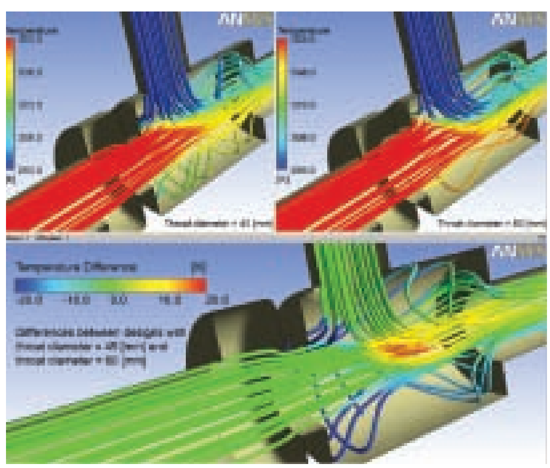
Conclusion

This is only a sampling of what the fluid dynamics development teams have produced for ANSYS 12.0. The combined depth and breadth of CFD knowledge and experience is delivering benefits to all users as technologies are combined and development teams drive simulation technology to new levels of achievement. With release 12.0, ANSYS continues its commitment to provide leading-edge CFD technology. ■

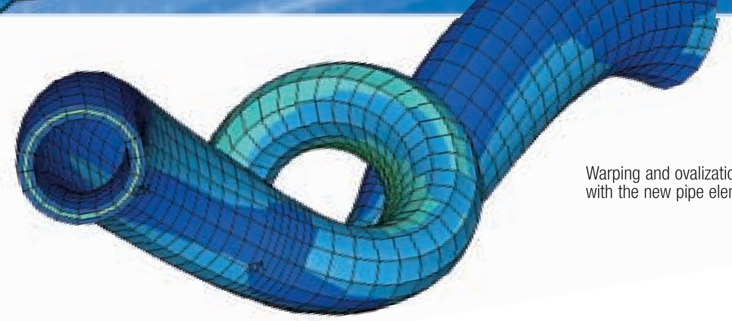
This article was written through contributions from Chris Wolfe and John Stokes of ANSYS, Inc.



In work sponsored by BMT Seatech, partially-filled tanks on marine vessels are being simulated by researchers at the University of Southampton to predict structural loads and changes in vessel behavior due to the sloshing of the fluid.



CFD-Post can be used to compare multiple designs directly, both by examining them side by side and by looking at the calculated difference between results.
Geometry courtesy CADFEM GmbH.



Warping and ovalization of pipe structures with the new pipe elements

Designing with **Structure**

Advancements in structural mechanics allow more efficient and higher-fidelity modeling of complex structural phenomena.

The ability to drive the engineering design process in structural applications has taken a significant step forward with the improvements in release 12.0. New features and tools, many integrated into the ANSYS Workbench platform, help reduce overall solution time. Specific improvements focus on elements, materials and contact and solver performance, along with linear, rigid and flexible dynamics.

Elements

The most notable new element in release 12.0 is the four-noded tetrahedron for modeling complex geometries in hyperelastic or forming applications. The element provides a convenient way to automate the meshing of complex structures, avoiding the need for pure hexahedral meshes. This reduces the time it takes to develop a case from geometry through solution, while maintaining the accuracy of the solution. See the table below for a summary of new and enhanced elements.

When simulating a nonlinear process, large deformation can introduce too much distortion of the elements. Resolving this requires local remeshing during the simulation process. The 2-D rezoning introduced with release 11.0 extends further in ANSYS 12.0, increasing the flexibility of the remeshing process: The user can now define transition regions within the refined zones and use meshes created in external meshing tools.

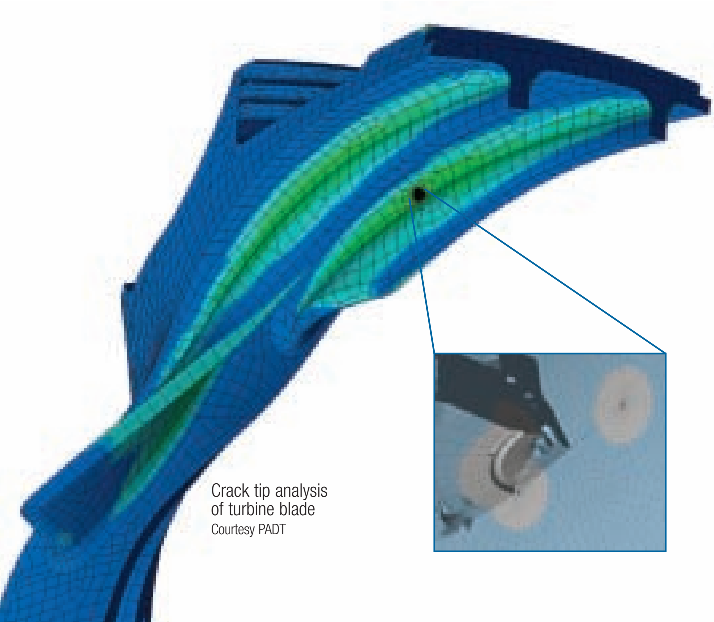
Materials

Accounting for proper cyclic softening or hardening or damage of materials is a key factor for elastomer applications and, more generally speaking, any structure whose material variation depends on the strain rate. Release 12.0 introduces several additions to the wide choice of materials already available. Other feature improvements include:

- Rate-dependent Chaboche plasticity, which can benefit turbine and engine design
- Bergström–Boyce model to enhance elastomer modeling capabilities
- New damage model based on the Ogden–Roxburgh formulation

Element	New	Improved	Capability	Applications
Four-noded tetrahedron	X		Provides a convenient way to automate meshing of complex structures, avoiding need for pure hexahedral meshes	Modeling complex geometries for forming or hyperelastic applications
General axisymmetric element	X		Supports contact	Compatible with 3-D non-axisymmetric loading and can use arbitrary axis of rotation
Various pipe model elements	X		Increased accuracy	To provide refined behavior of structures in case of ovalization, warping or similar deformations of cross section for thin or moderately thick pipes and nonlinear material behavior support
Shell: linear, quadratic, axisymmetric		X	Improved shell thickness updating scheme and improved convergence	Provides greater accuracy in the behavior of shell models as well as a faster solution for nonlinear problems
Beam		X	Supports cubic shape function	Provides additional accuracy to coarse meshes and greater support of complex load patterns
Reinforcement elements		X	Allows modeling of discrete fibers with a variety of nonlinear material behavior	Stresses in reinforcements can be analyzed separately from host elements

Summary of new and enhanced element features in ANSYS 12.0 structural analysis products



Crack tip analysis of turbine blade
Courtesy PADT

- Anand's viscoplasticity model, useful for metal forming applications such as solder joints
- Improvements in the calculation of J-integrals to account for mixed-mode stress intensity factors, which benefit improvements in fracture mechanics
- Initial strain and initial plastic stress import capabilities that allow for state transfer from a 2-D model to a 3-D model

Contact

As assemblies have become a de facto standard in simulation, the need for advanced contact features has grown accordingly. ANSYS 12.0 developments include a number of additional contact modeling features as well as significant improvements in solving contact problems.

While Coulomb's law for friction is widely used, there are circumstances in which more elaborate modeling is required, such as wear modeling or pipelines resting on sea beds. Release 12.0 supports a friction coefficient definition that depends upon the contact state itself and accounts for complex frictional behavior. Specifically, the user is able to define the dependency of the friction on contact parameters, such as sliding distance or contact pressure.

A typical contact application involves seals that are subject to fluid pressure. Release 12.0 provides support of fluid pressure penetration, to model scenarios in which pressure rises higher than the contact pressure around the seal. Pressures in such cases can be applied only on the free faces of the structure and evolve with the contact state.

Contact simulation is usually a time-consuming process. The latest release introduces contact modeling improvements that significantly reduce computation time and results file size. These enhancements include new contact search algorithms, contact trimming logic and smart over-constraint elimination for multipoint constraint (MPC) contact.

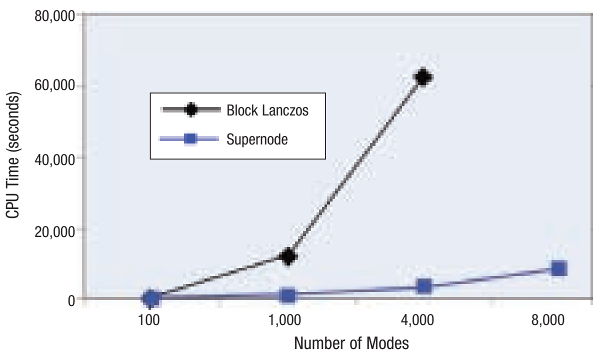
Solver Performance

Solver performance has improved in many different areas. ANSYS 12.0 introduces a new modal solver, called SNODE, that increases the speed of computation for problems with a large number of modes — in the realm of several hundred — on large structures that typically have over a million degrees of freedom. This solver is well suited for automotive or aerospace applications and for large beams and shell assemblies. Beyond its ability to compute a larger number of modes in a reduced amount of time, SNODE also significantly reduces the amount of I/O required to compute the solution. (See the Supernode Eigensolver article.)

Many enhancements have been made to the distributed solver to improve the scalability of the solution. (See the article on High Performance Computing.) More solver techniques are supported, including:

- Partial solve capability that computes only a portion of the solution
- Prestressed analysis
- Models that employ the use of unsymmetric matrices, which are useful for scenarios that involve high-friction coefficients, for example

These new features can be combined for applications such as brake squeal, which might combine the partial solve and unsymmetric matrix capabilities.

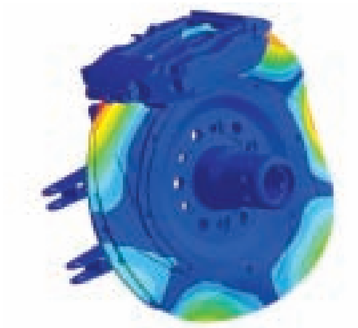


Performance of new modal solver

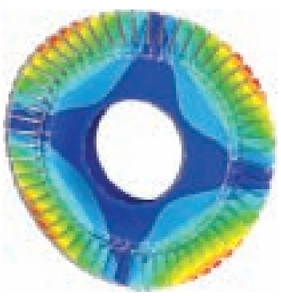
Linear Dynamics

Some of these element, material, contact and solver improvements benefit the field of linear dynamics as well. They are complemented by enhancements specific to this simulation area, especially for mode superposition analysis. For harmonic or transient loadings, the mode superposition methods exhibit better performance, especially during the so-called expansion pass that computes results at each frequency or time step on the full model. For very large structures, the total computation effort can be reduced by up to 75 percent. The mode combination for spectral analysis benefits from similar advancements. Instability predictions, such as the case of brake squeal, can be computed faster due to several enhancements to the damped eigensolver.

The introduction of ANSYS Variational Technology provides faster mode computation for cyclic symmetric structures, such as those found in many turbine applications. Using this technique can typically improve solution speed by a factor of three or four — the greater the number of sectors, the better the performance.



Instability analysis for brake squeal



Modal analysis of a cyclic–symmetric geometry
Courtesy PADT, Inc.

Rotating machinery applications profit from an extended set of capabilities for rotordynamics analysis. These include the extension of the gyroscopic effect to shell and 2-D elements and inclusion of rotating damping that takes hysteretic behavior into account.

Random vibration and spectral analysis users gain new tools as well as a greater flexibility in modeling structures, including support of spectrum analysis in the ANSYS Workbench platform. New tools include the United States Nuclear Regulatory Commission–compliant computation of missing masses and support of rigid modes, along with the ability to use residual vectors to account for higher energy modes. The global number of spectra applied simultaneously to the structure has been increased up to 50 as has the number of modes used in a combination — now up to 10,000.

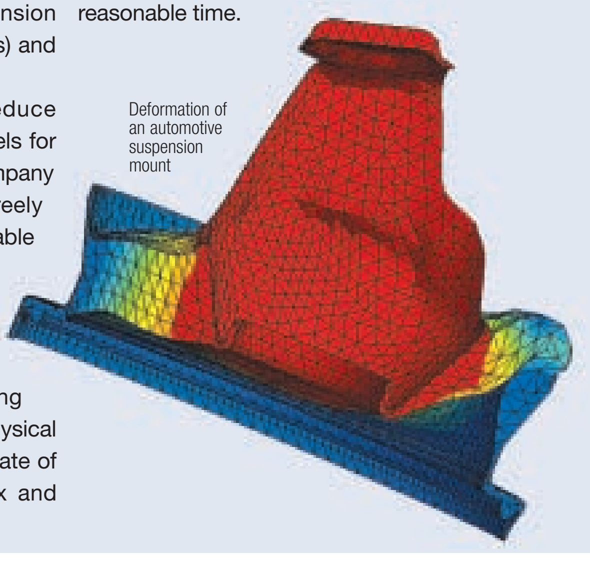
When analyzing design variations, comparing data from different simulation cases, or correlating simulation and test data, comparison between modal content of the models is required. The modal assurance criterion (MAC) in release 12.0 provides a convenient tool to compare the results of two modal analyses. Typical use cases for the criteria include tuning of misaligned turbine blades or validation of new component designs, each with respect to their vibration behavior.

New Element Reduces Meshing Time

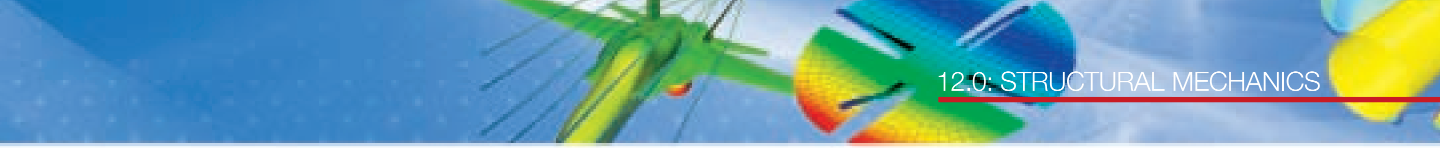
ZF Boge Elastmetall GmbH develops, manufactures and supplies vibration control components and parts for the automotive industry. These components include plastic parts, energy-absorbing elements for vehicle safety, and rubber–metal components such as chassis suspension mounts, control arm bushes (also known as bushings) and engine mounts.

The German company uses simulation to reduce development time and costs. When developing models for components with hyperelastic material properties, company engineers require an element type that can be freely meshed; can accommodate extreme deformation, stable contact and short computing time; and can provide reliable results.

By using the new SOLID285 four-noded tetrahedron element available in ANSYS 12.0, ZF Boge Elastmetall engineers considerably reduced meshing time. Close correlation between the simulation and physical measurement allowed them to determine the spring rate of strongly deformed structures without the complex and time-consuming meshing that was previously required when using hexahedral elements. Boge's work proved that by employing this new element, users can determine the stresses and strains for a durability calculation in a reasonable time.



Deformation of an automotive suspension mount



ANSYS Workbench Integration

The integration of the structural applications within the ANSYS Workbench platform provides additional productivity to users, including:

- New meshing techniques to improve mesh quality
- Support of additional elements, such as gasket elements as well as quadratic shells and beams that include offset definitions
- Boundary condition definitions that provide a spatial dependency for loads
- Coupling conditions
- Remote points
- Ability to associate contact to the top or bottom of shell face

Post-processing capabilities have drastically improved with release 12.0. The user can now plot any structural simulation data stored in the results files. Mathematical operations involving elementary results can be introduced to create additional user-defined criteria. Complex mode shapes, plotting on linear paths, stress linearization (which depends upon path plotting), and the ability to display unaveraged results at element nodes complement the list of the features that increase productivity at ANSYS 12.0. ■

Pierre Thieffry and Siddharth Shah of ANSYS, Inc. contributed to this article.

Multibody Dynamics

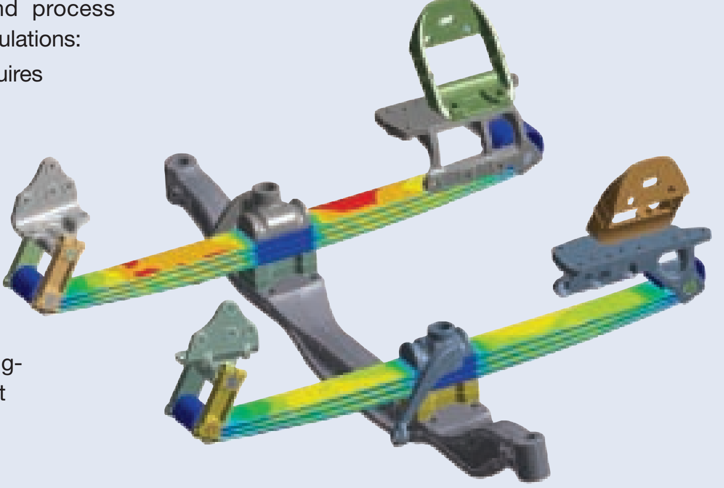
At release 12.0, a number of improvements in the general area of multibody dynamics enable the rapid design and analysis of complete mechanical systems undergoing large overall motion. ANSYS Rigid Dynamics software has a new Runge–Kutta 5 integrator, the preferred solution for long transient simulations. A new bushing joint, a "stops and locks" option for most other joint types, and the ability to specify preload for springs give new flexibility when simulating complex multiple-part assemblies and component interactions.

For complex assemblies, conducting an initial simulation with the ANSYS Rigid Dynamics product is the key to achieving robust flexible dynamics results. Creating over-constrained assemblies is an inconvenient reality; release 12.0 adds a redundancy analysis and repair tool to identify overconstrained assemblies, points out which joints or degrees of freedom are redundant, and allows selective unconstraining to create a properly constrained mechanism.

A number of improvements to data and process handling increase ease of use for multibody simulations:

- Enhanced load data fitting (no longer requires curve fitting)
- Ability to read in complex load input, such as simulated or measured multi-channel road surface or seismic data, and apply as load data to parts or joints
- Ability to use remote solution manager (RSM) to offload the solving effort to a server or other capable CPU (benefits long-duration and multi-channel input transient simulations)
- Ability to export forces and moments at any time within a transient simulation

For durability studies, exported loads can be used in a static structural analysis as an efficient first-pass failure analysis. Although it won't provide the complete picture obtained from comprehensive flexible dynamics simulation, a static structural simulation is typically much less computationally expensive. Flexible dynamics simulations benefit at release 12.0 from robust component modal synthesis, or CMS. This method uses an internal substructuring approach and requires that the CMS parts of an assembly are constructed with linear materials. The procedure simplifies a problem by accounting only for a few degrees of freedom, which results in solution times that are often a fraction of those found using the standard full computation method. Time-to-solution reductions of several hundred percent are not uncommon.



Multibody dynamics capabilities were used to simulate this leaf spring suspension.

Explicit Dynamics Goes Mainstream

ANSYS 12.0 brings native explicit dynamics to ANSYS Workbench and provides the easiest explicit software for nonlinear dynamics.

ANSYS has expended significant effort in the area of explicit dynamics for release 12.0 — including the addition of a new product that will make this technology accessible to users independent of their simulation experience. In addition, enhancements to both the ANSYS AUTODYN and ANSYS LS-DYNA products provide considerable benefits to their users.

Newly introduced in ANSYS 12.0, ANSYS Explicit STR software is the first explicit dynamics product with a native ANSYS Workbench interface. It is based on the Lagrangian portion of the ANSYS AUTODYN product. The technology will appeal to those who want to model transient dynamic events such as drop tests, as well as quasi-static events involving rapidly changing contact conditions, sophisticated material failure/damage and/or severe displacements and rotations of structures. In addition, it will appeal to users who can benefit from the productivity provided by other applications integrated within the ANSYS Workbench environment. Those who have previous experience using ANSYS Workbench will find that they already know most of what is needed to use ANSYS Explicit STR.

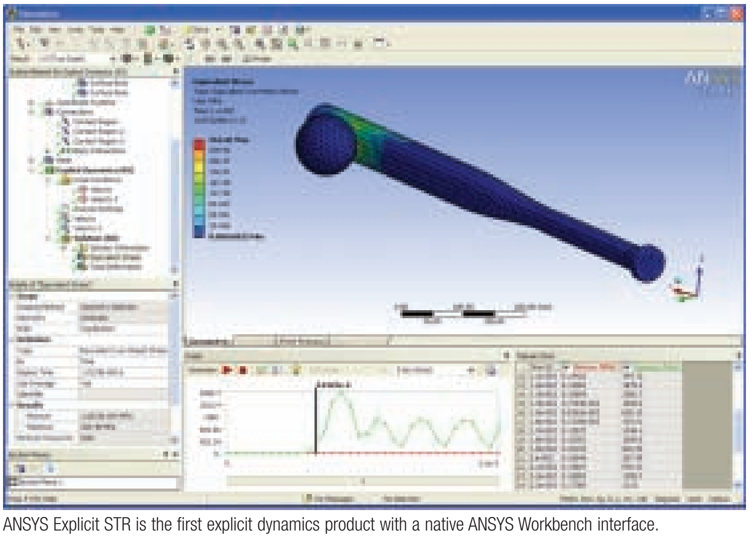
The ANSYS Explicit STR tool is well suited to solving:

- Drop tests (electronics and consumer goods)
- Low- to high-speed solid-to-solid impacts (a wide range of applications from sporting goods to aerospace)
- Highly nonlinear plastic buckling events (for ultimate limit state design)
- Complete material failure applications (defense and homeland security)
- Breakable contact, such as adhesives or spot welds (electronics and automotive)

The real benefit of ANSYS Explicit STR software is the work flow afforded by operating in the ANSYS Workbench environment. While many different simulation processes are possible, here is an example of the typical steps a user might take:

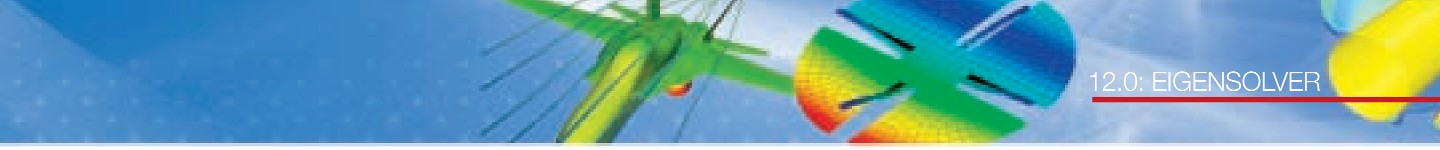
- Associatively link to a parametric CAD model or import a geometry
- Create a smooth explicit mesh using the new explicit preference option and/or patch-independent mesh method within the ANSYS meshing platform; automatically create part-to-part contact by using the new body interactions tool
- Fine-tune contact specifications if desired by utilizing breakable or eroding contact options
- Load and/or support an assembly and/or parts as usual
- Assign material properties from the comprehensive material library
- Solve interactively either in the background or via remote solution manager (RSM)
- View progress of solution in real time using concurrent post-processing capability, new to ANSYS Workbench at 12.0
- Explore alternative design ideas via parametric changes to the CAD model and easily perform re-solves, just like other ANSYS Workbench based applications
- Use the ANSYS Design Exploration capability to automate the parametric model space exploration

In addition, users of the full version of ANSYS AUTODYN (structural- plus fluids-capable) have access to the ANSYS Explicit STR interface; consequently, they will be able to transfer implicit solutions from the ANSYS Workbench environment for doing implicit–explicit solutions, such as bird strike analysis of a pre-stressed fan blade. ANSYS LS-DYNA software users will be able to use the pre-processing portion of ANSYS Explicit STR and output a .K file for solving and post-processing outside of ANSYS Workbench. ■



ANSYS Explicit STR is the first explicit dynamics product with a native ANSYS Workbench interface.

Wim J. Slagter of ANSYS, Inc. is available to answer your questions about explicit dynamics.



Introducing the Supernode Eigensolver

A new eigensolver in ANSYS 12.0 determines large numbers of natural frequency modes more quickly and efficiently than conventional methods.

By Jeff Beisheim, Senior Development Engineer, ANSYS, Inc.

In a wide range of applications, parts are subject to cyclic mechanical loading, and engineers must use an eigensolver to determine the structure's natural frequencies — also known as eigen modes. With some modes, large vibration amplitudes can interfere with product performance and cause damage, such as fatigue cracking. In most cases, only the first few modes with the largest deformations are of particular interest, though determining even dozens of modes can be common.

In the CAE industry, the block Lanczos eigensolver is typically used more than any other for these types of calculations. This proven algorithm has been used in many finite element software packages, including ANSYS Mechanical technology. It brings together the efficiency and accuracy of the Lanczos algorithm and the robustness of a sparse direct equation solver. The software works in a sequential fashion by computing one mode (or a block of modes) at a time until all desired modes have been computed.

Although the method is considered efficient in solving for each of these eigen modes, the amount of time and computer resources (both memory and I/O) required adds up when many dozens of eigen modes must be found. Elapsed solution times of several hours — or days — are typical in applications that involve thousands of modes. Generally, determining large numbers of modes is required in capturing system response for studies such as transient or harmonic analyses using the mode superposition method.

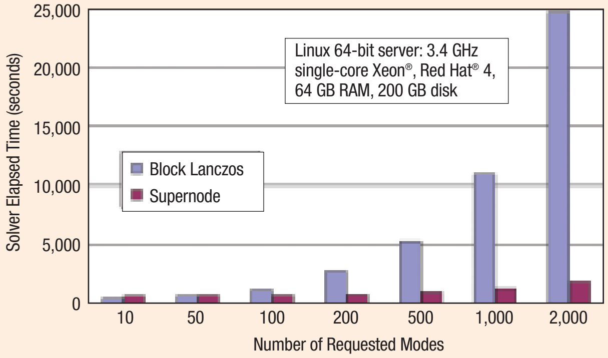
The ANSYS supernode eigensolver is well suited for applications such as seismic analysis of power plant cooling towers, skyscrapers and other structures in which hundreds of modes must be extracted to determine the response of the structures to multiple short-duration transient shock/impact loadings.

For such cases, the ANSYS release 12.0 includes a new supernode eigensolver. Instead of computing each mode individually and working with mode shapes in the global model space, the supernode algorithm uses a mathematical approach based on substructuring to simultaneously determine all modes within a given frequency range and to manage data in a reduced model space.

By utilizing fewer resources than block Lanczos, this supernode eigensolver becomes an ideal choice when solving on a desktop computer, which can have limited memory and relatively slow I/O performance. When combined with current eigensolver technology already available in mechanical software from ANSYS, virtually all modal analyses can be efficiently solved.

Comparing Eigensolvers

A sample comparison shows that the supernode eigensolver offers no significant performance advantage over block Lanczos for a low number of modes. In fact, supernode is slower when 50 or fewer modes are requested. However, when more than 200 modes are requested, the supernode eigensolver is significantly faster than block Lanczos — with efficiency increasing considerably as the number rises.



Performance of block Lanczos and supernode eigensolvers at 1 million DOF

Using Supernode Eigensolver

The supernode eigensolver can be selected in the ANSYS Mechanical traditional interface using the `SNODE` label with the `MODOPT` command or via the Analysis Options dialog box. ANSYS Workbench users can choose this eigensolver by adding a command snippet that includes the `MODOPT,SNODE` command.

The `MODOPT` command allows users to specify the number of natural frequencies and what range those frequencies lie within. With other eigensolvers, the number of requested modes primarily affects solver performance, while the frequency range is, essentially, optional. Asking for more modes increases solution time, while the frequency range generally decides which computed modes are computed.

The supernode eigensolver behaves completely opposite: It computes all modes within the specified frequency range regardless of how many modes are requested. Therefore, for maximum efficiency, users should input a range that covers only the spectrum of frequencies between the first and last mode of interest. The number of modes requested on the `MODOPT` command then decides how many of the computed frequencies are provided by the software.

Today, with the prevalence of multi-core processors, the first release of this new eigensolver will support shared-memory parallelism. For users who want full control of the solver, a new `SNOPTION` command allows control over several important parameters that affect accuracy and efficiency.

Controlling Parameters

The supernode eigensolver does not compute exact eigenvalues. Typically, this is not an issue, since the lowest modes in the system (often used to compute the dominant resonant frequencies) are computed very accurately — generally within less than 1 percent compared to using block Lanczos. Accuracy drifts somewhat with higher modes, however, in which computed values may be off by as much as a few percent compared with Lanczos. In these cases, the accuracy of the solver may be tightened using the range factor (`RangeFact`) field on the `SNOPTION` command. Higher values of `RangeFact` lead to more accurate solutions at the cost of extra computations that somewhat slow down eigensolver performance.

When computing the final mode shapes, the supernode eigensolver often does the bulk of I/O transfer to and from disk, and the amount of I/O transfer is often significantly less than a similar run using block Lanczos. To maximize supernode solver efficiency, I/O can be further minimized using the block size (`BlockSize`) field on the `SNOPTION` command. Larger values of block size will reduce the amount of I/O transfer by holding more data in memory during the eigenvalue/eigenvector output phase, which generally speeds up the overall solution time. However, this is recommended only if there is enough physical memory to do so.

Application Guidelines

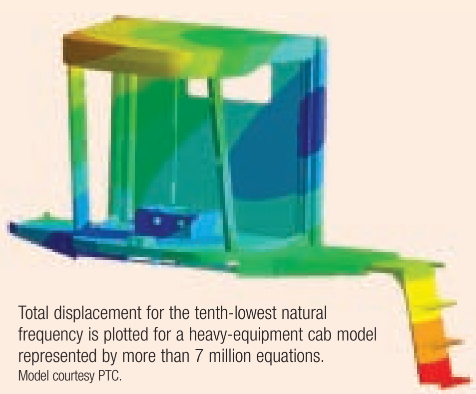
The following general guidelines can be used in determining when to use the supernode eigensolver, which is typically most efficient when the following three conditions are met:

- The model would be a good candidate for using the sparse solver in a similar static or full transient analysis (that is, dominated with beam/shell elements or having thin structure).
- The number of requested modes is greater than 200.
- The beginning frequency input on the `MODOPT` command is zero (or near zero).

For models that have dominantly solid elements or bulky geometry, the supernode eigensolver can be more efficient than other eigensolvers, but it may require higher numbers of modes to consider it the best choice. Also, other factors such as computing hardware can affect the decision. For example, on machines with slow I/O performance, the supernode eigensolver may be the better choice, even when solving for less than 200 modes. ■

Examining Real-World Performance

A heavy-equipment cab model with over 7 million equations was used to demonstrate the power of the supernode eigensolver. This model was solved using a single core on a machine with the Windows® 64-bit operating system with 32 gigabytes of RAM. Time spent computing 300 modes with block Lanczos was about 31.8 hours. The solution time dropped to 15.7 hours (a two-times speedup) using the supernode eigensolver. The model illustrates real-world performance for a bulkier model with only 300 modes requested. For modal analyses in which hundreds or thousands of modes are requested, users often see a speedup of 10 times or more with the supernode eigensolver compared with block Lanczos. In one recent project, a major industrial equipment manufacturer reduced analysis run time from 1.5 hours to just 10 minutes by switching from block Lanczos to supernode eigensolver.



Total displacement for the tenth-lowest natural frequency is plotted for a heavy-equipment cab model represented by more than 7 million equations.
Model courtesy PTC.

The Need for **Speed**

From desktop to supercomputer, high-performance computing with ANSYS 12.0 continues to race ahead.

Tuning software from ANSYS on the latest high-performance computing technologies for optimal performance has been — and will continue to be — a major focus area within the software development organization at ANSYS. This effort has yielded significant performance gains and new functionality in ANSYS 12.0, with important implications for more productive use of simulation by customers.

High-performance computing, or HPC, refers to the use of high-speed processors (CPUs) and related technologies to solve computationally intensive problems. In recent years, HPC has become much more widely available and affordable, primarily due to the use of multiple low-cost processors that work in parallel on the computational task. Today, clusters of affordable compute servers make large-scale parallel processing a very viable strategy for ANSYS customers. In fact, the new multi-core processors have turned even desktop workstations into high-performance platforms for single-job execution.

This wider availability of HPC systems is enabling important trends in engineering simulation. Simulation models are getting *larger* — using more computer memory and requiring more computational time — as engineers include greater geometric detail and more-realistic treatment of physical phenomena (Figure 1). These higher-fidelity models are critical for simulation to reduce the need for expensive physical testing. HPC systems make higher-fidelity simulations practical by yielding results within the engineering project's required time frame. A second important trend is toward *more* simulations — enabling engineers to consider multiple design ideas, conduct parametric studies and even perform automated design optimization. HPC systems provide the throughput required for completing multiple simulations simultaneously, thus allowing design decisions to be made early in the project.

Software from ANSYS takes advantage of multi-processor and/or multi-core systems by employing domain decomposition, which divides the simulation model into multiple pieces or sub-domains. Each sub-domain is then computed on a separate processor (or core), and the multiple processors work in parallel to speed up the computation. In the ideal case, speedup is linear, meaning that the simulation turnaround time can be reduced in proportion to the number of processors used. Parallel processing also allows larger problems to be tackled, since the processing power and memory requirements can be distributed across the cluster of processors. Whether performed on a multi-core desktop workstation, desk-side cluster or scaled-out HPC system, parallel

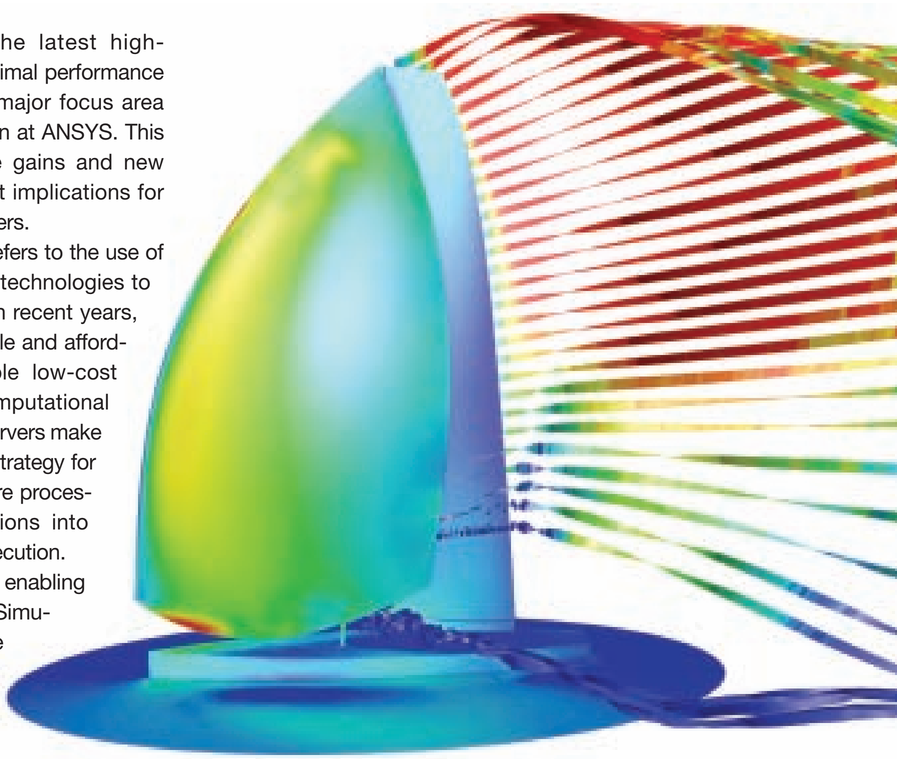


Figure 1. Simulations as large as 1 billion cells are now supported at release 12.0. This 1 billion-scale racing yacht simulation was conducted on a cluster of 208 HP ProLiant™ server blades. (For more information, visit www.ansys.com/one-billion.)
Image courtesy Ignazio Maria Viola.

HPC on Workstations?

While purists might argue whether workstations can be considered high-performance computing platforms, the performance possibilities for ANSYS 12.0 running on workstations are noteworthy. With the latest quad-core processor technology, an eight-core workstation running Windows® can deliver a speedup of five to six times for users of mechanical products from ANSYS (Figure 2) and over seven times for users of its fluid dynamics products (Figure 4). This means that parallel processing now provides tremendous ROI for both large engineering groups and individual workstation users, enabling faster turnaround, higher-fidelity models and parametric modeling. With release 12.0 and 2009 computing platforms, parallel processing improves productivity for all simulation types, from workstation to cluster, for mechanical or fluids simulations.

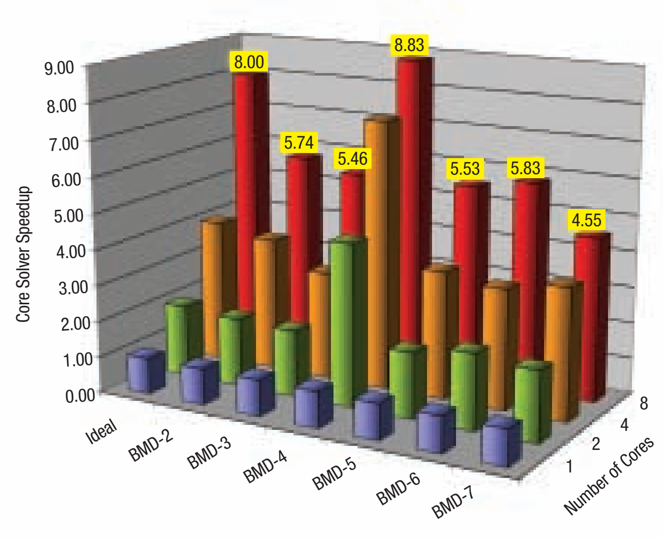


Figure 2. Speedup of Distributed ANSYS Mechanical 12.0 software using the 11.0 SP1 benchmark problems. Simulations running eight-way parallel show typical speedup of between five and six times. Data was collected on a Cray CX-1 Personal Supercomputer using two quad-core Intel Xeon Processor E5472 running Microsoft® Windows HPC Server 2008.

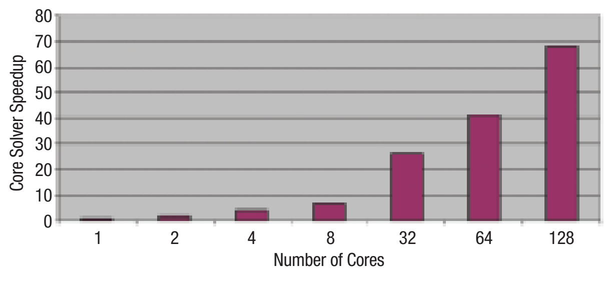


Figure 3. Scaling of a 10M DOF simulation using the ANSYS Mechanical 12.0 iterative PCG solver on a cluster of Intel Xeon 5500 Processor series. All cores on these quad-core processors are fully utilized for the benchmark.

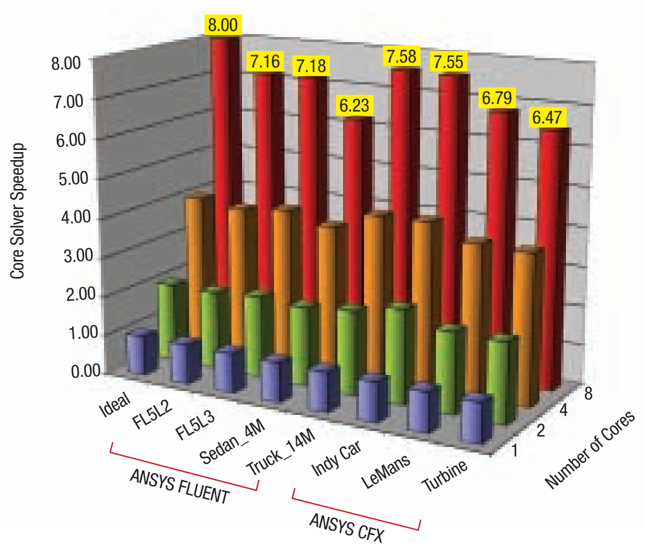


Figure 4. Scalability of ANSYS FLUENT and ANSYS CFX benchmark problems on the Intel Xeon 5500 Processor series quad-core platform. Simulations running eight-way parallel show typical speedup of over seven times.

processing provides excellent return on investment by improving the productivity of the engineers who perform simulation.

ANSYS 12.0 provides many important advances in areas related to parallel processing and HPC, delivering scalability from desktop systems to supercomputers. For users of the ANSYS Mechanical product line, release 12.0 introduces expanded functionality in the Distributed ANSYS (DANSYS) solvers, including support for all multi-field simulations, pre-stress effects and analyses involving cyclic symmetry. In addition, DANSYS now supports both symmetric and non-symmetric matrices as well as all electromagnetic analyses. Mechanical simulations benefit from significantly improved scaling on the latest multi-core processors. Simulations in the size range of 2 million to 3 million degrees of freedom (DOF) now show good scaling on eight cores (Figure 2). Based on benchmark problem performance, customers can expect to get answers back five to six times faster on eight cores. Even more impressive is the scale-out behavior shown in Figure 3, with a 10 million DOF simulation showing solver speedup of 68 times on 128 cores.

With turnaround times measured in tens of seconds, parametric studies and automated design optimization are now well within the grasp of ANSYS customers who perform mechanical simulations. These benchmarks are noteworthy, in part, as they show execution with all cores on the cluster fully utilized, indicating that the latest quad-core processors have sufficient memory bandwidth to support parallel processing for memory-hungry mechanical simulations. Software tuning has contributed to improved scaling as well, including improved domain decomposition, load balancing and distributed matrix generation. To help customers maximize their ANSYS solver performance, the online help system now includes a performance guide that provides a comprehensive summary of factors that impact the performance of mechanical simulations on current hardware systems.

Explicit simulations using ANSYS AUTODYN technology take great advantage of HPC systems at release 12.0. Full 64-bit support is now available, allowing much larger simulations to be considered from pre-processing to solution and post-processing.

For users of fluid dynamics software from ANSYS, release 12.0 builds on the strong foundation of excellent scaling in both the ANSYS FLUENT and ANSYS CFX solvers. These fluids simulation codes run massively parallel, with sustained scaling at hundreds or even thousands of cores. The release incorporates tuning for the latest multi-core processors, including enhanced cache re-utilization, optimal mapping and binding of processes to cores (for better memory locality and system utilization), and leveraging the latest compiler optimizations. The resulting ANSYS FLUENT and ANSYS CFX performance on the newly released Intel® Xeon® 5500 Processor series is shown in Figure 4, with outstanding speedup of over seven times for many benchmark cases. In addition, the new release delivers significant performance improvements at large core counts, the result of general solver enhancements and optimized communications over the latest high-speed interconnects. Figure 5 demonstrates scaling achieved by ANSYS CFX software on a cluster of quad-core AMD processors. Nearly ideal linear scaling to 1,024 cores — and very good efficiency up to 2,048 cores — has been demonstrated with ANSYS FLUENT (Figure 6). Both fluids codes provide improvements to mesh partitioning that enhance scalability. ANSYS FLUENT software now provides dynamic load balancing based on mesh- and solution-derived criteria. This enables optimal scalability for simulations involving multiphysics, such as particle-laden flows. The ANSYS CFX code delivers improved partitioning for moving and/or rotating meshes, yielding important reductions in memory use and improved performance for turbomachinery and related applications. Finally, ANSYS FLUENT users will benefit from several usability improvements, including built-in tools for checking system network bandwidth, latency and resource utilization — all helping to identify potential scaling bottlenecks on the cluster.

Beyond solver speedup, the ANSYS 12.0 focus on HPC addresses issues related to file input and output (I/O). Both ANSYS FLUENT and ANSYS CFX software have updated I/O algorithms to speed up writing of results files on clusters, enhancing the practicality of periodic solution snapshots when checkpointing or running time-dependent simulations. ANSYS FLUENT includes improvements in the standard file I/O as well as new support for fully parallel I/O based on parallel file systems. Order of magnitude improvements in I/O throughput have been demonstrated on large test cases (Figure 7), virtually eliminating I/O as a potential bottleneck for large-scale simulations. ANSYS CFX improves I/O performance via data compression during the process of gathering from the cluster nodes, therefore reducing file write times. Proper I/O configuration is also an important aspect of cluster performance for the ANSYS Mechanical product line.

Recognizing that cluster deployment and management are key concerns, ANSYS 12.0 includes a focus on compatibility with the overall HPC ecosystem. ANSYS products are registered and tested as part of the Intel Cluster Ready program, confirming that these products conform to standards of compatibility that contribute to successful deployment (www.ansys.com/intelclusterready). In addition to supporting enterprise Linux® distributions from Red Hat® and Novell, ANSYS 12.0 products are supported on clusters based on Microsoft Windows HPC Server 2008. ANSYS has also worked with hardware OEMs, including HP®, SGI®, IBM®, Dell®, Cray® and others, to define reference configurations that are optimally designed to run simulation software from ANSYS (www.ansys.com/reference-configs).

As computing technology continues to evolve, ANSYS is working with HPC leaders to ensure support for the breakthrough capability that will make simulation more productive. Looking forward, important emerging technologies include many-core processors, general purpose graphical processing units (GP-GPUs) and fault tolerance at large scale. ■

Contributions to this article were made by Barbara Hutchings, Ray Browell and Prasad Alavilli of ANSYS, Inc.

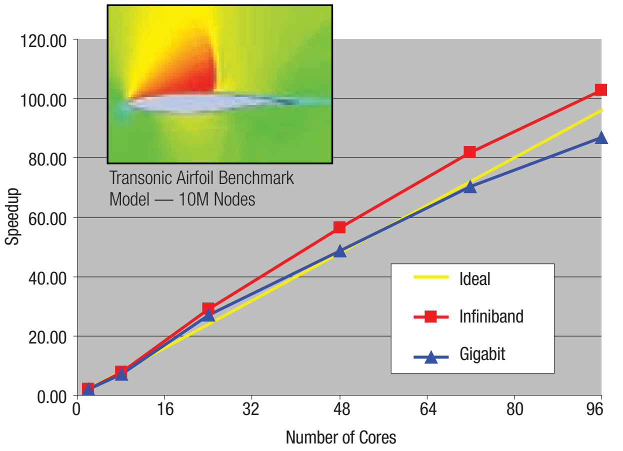


Figure 5. Scalability of ANSYS CFX 12.0 on a 10M node transonic airfoil benchmark example. Data was collected on a cluster of AMD Opteron™ 2218 processors, showing the benefit of a high-speed interconnect.

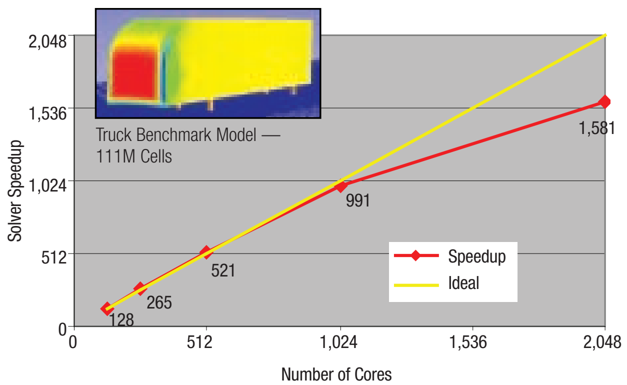


Figure 6. Scaling of ANSYS FLUENT 12.0 software is nearly ideal up to 1,024 processors and 78 percent of ideal at 2,048 processors. Data courtesy SGI, based on the SGI Altix® ICE 8200EX using quad-core Intel Xeon Processor E5472 with Infiniband®.

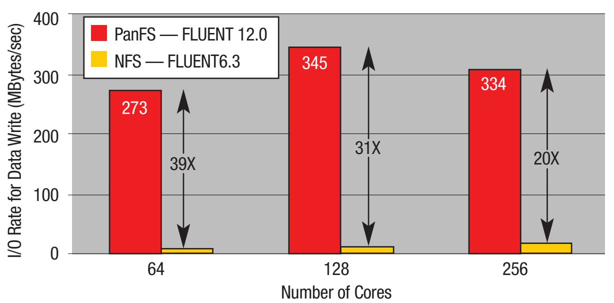


Figure 7. Parallel I/O in ANSYS FLUENT 12.0 using the Panasas© file system, compared to serial I/O in the previous release using NFS. Parallel treatment of I/O provides important speedup for time-varying simulations on large clusters.

Foundations for the Future

The many advanced features of ANSYS 12.0 were designed to solve today's challenging engineering problems and to deliver a platform for tomorrow's simulation technology.

As this special spotlight in *ANSYS Advantage* attests, release 12.0 delivers a compelling advancement in what the CAE industry has, until now, only envisioned — a full range of best-in-class simulation capabilities assembled into a flexible multiphysics simulation environment specifically designed to increase engineering insight, productivity and innovation. Whether the need is structural analysis, fluid flow, thermal, electromagnetics, geometry preparation or meshing, ANSYS customers can rely on release 12.0 for the depth and breadth of simulation capabilities to overcome their engineering challenges.

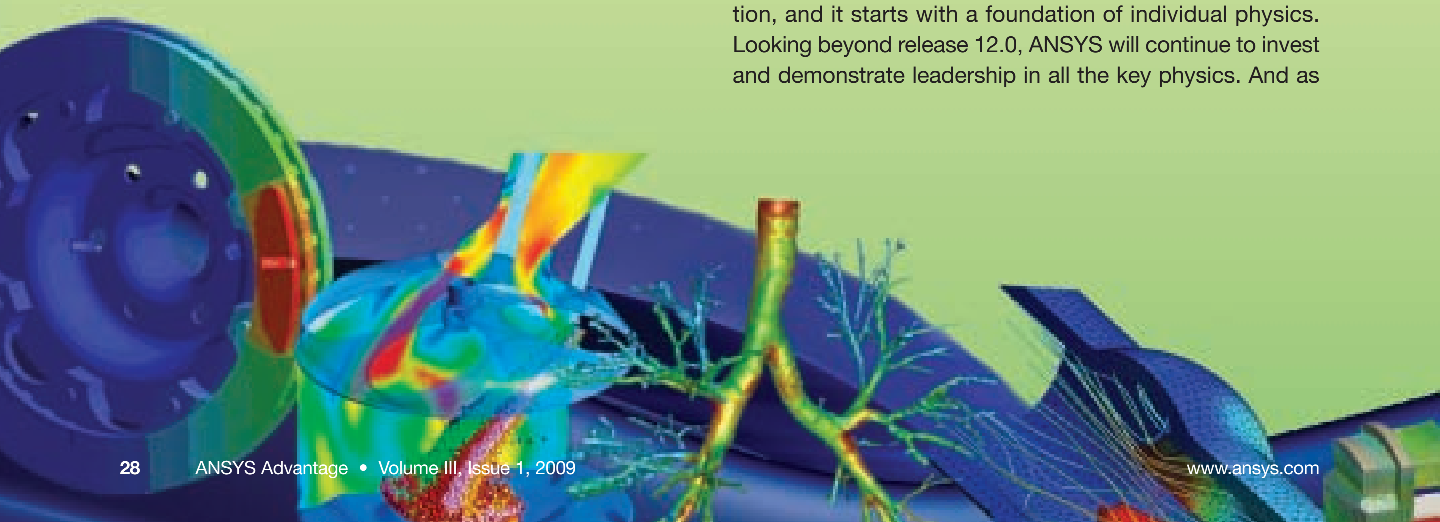
Staying true to our commitment to develop the most advanced simulation technologies, release 12.0 has further expanded the depth of individual physics and more intimately coupled them to form an engineering simulation capability second to none. A multitude of new material models, physics and algorithms enable simulating real-world operating conditions and coupled physical phenomena, while new solver technology and parallel processing improvements have dramatically reduced run times and made complete system simulations more computationally affordable.

Shouldering the array of technology in release 12.0 is our next-generation simulation platform, ANSYS Workbench 2.0. Seamlessly spanning all stages of engineering simulation, ANSYS Workbench 2.0 has been engineered to manage the complexities of today's simulations and to accelerate innovation.

Release 12.0 is a notable milestone in the company's nearly 40-year history of innovating engineering simulation, and it sets the stage for a new era of Smart Engineering Simulation — an era in which ANSYS customers will gain more from their investment in simulation by increasing the efficiency of their processes, increasing the accuracy of their virtual prototypes, and capturing and reusing their simulation processes and data. However, the advancements of ANSYS 12.0 notwithstanding, the journey is far from complete. To address the simulation challenges on the horizon, ANSYS will continue to reinvest in research and development and to explore new technologies. In particular, there are a few areas that we consider vital in the pursuit of Simulation Driven Product Development — areas in which ANSYS has laid strong foundations and remains committed to build upon as we look beyond release 12.0.

Physics First

ANSYS customers rely heavily on simulation before making commitments to product designs or manufacturing processes. High-fidelity engineering simulation is absolutely paramount when upstream engineering decisions can determine the overall success of a product and, in some cases, the company's financial success. At ANSYS, we believe our customers should never have to compromise by making broad-based engineering assumptions due to limitations in their analysis software. That is why we have taken a comprehensive multiphysics approach to simulation, and it starts with a foundation of individual physics. Looking beyond release 12.0, ANSYS will continue to invest and demonstrate leadership in all the key physics. And as



we develop tomorrow's advanced capabilities, we will continue to allow them to be combined in ways that free engineers from making the assumptions associated with single-physics simulations. Within the ANSYS Workbench simulation paradigm, we will enable engineers to routinely consider the effects of fully coupled physical phenomena.

High-Performance Computing

As one might expect, high-performance computing (HPC) is a strategic enabling technology for ANSYS. The appearance of quad-core machines on the desktop and the increased availability of compute clusters have ushered in a new era of parallel and distributed computing for our customers. ANSYS has kept pace with the exponential increase in computational horsepower with prolific development in the areas of parallel and distributed computing and numerical methods. The result is improved scalability and dramatically reduced run times for large-scale fluid flow, structural and electromagnetic simulations.

Solving large-scale problems with meshes exceeding 1 billion cells has been the latest stretch goal for fluid flow simulation. Recently, HPC and software from ANSYS were combined to investigate the aerodynamics of a racing yacht using 1 billion computational cells. Breaking this barrier demonstrates our conviction for high-performance scientific computing. As computational resources increase and engineering simulations become larger and more complex, we will continue to ensure that our solvers scale appropriately. Moreover, our forward deployment of HPC technology is not limited to solvers. The complexity of today's models and massive amounts of results data require more-scalable solutions for preparing models and interpreting results as well.

ANSYS Workbench Framework

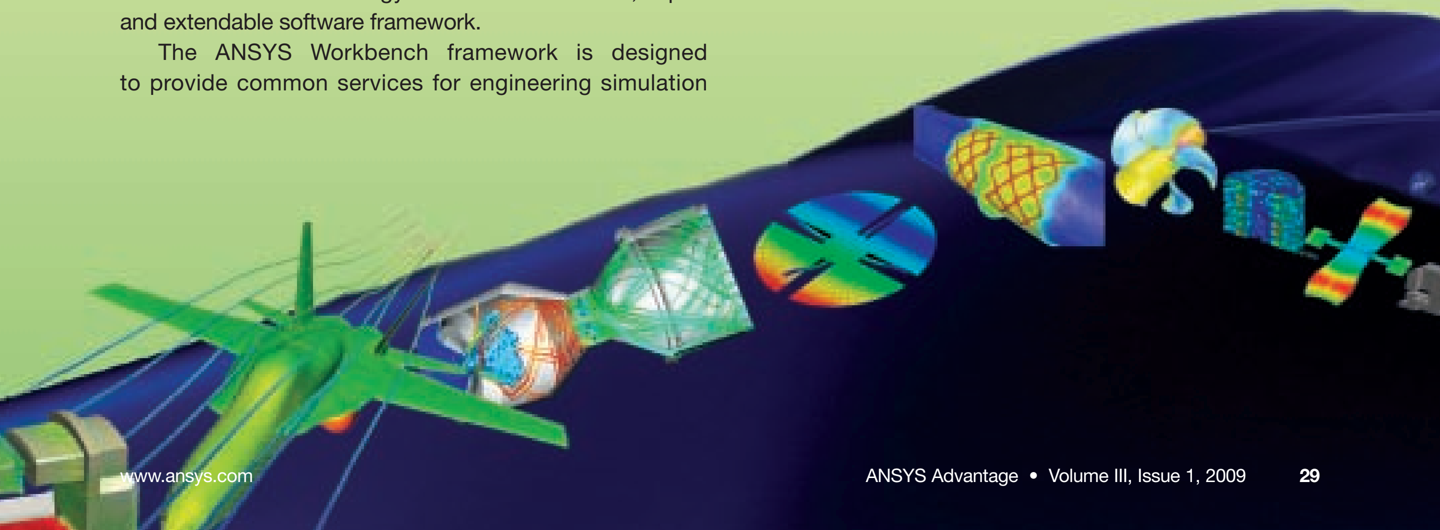
The ANSYS Workbench 2.0 platform is a powerful multi-domain simulation environment that harnesses the core physics from ANSYS; enables their interoperability; and provides common tools for interfacing with CAD, repairing geometry, creating meshes and post-processing results. Instrumental to the successful integration of this unparalleled breadth of technology is a "well-architected," open and extendable software framework.

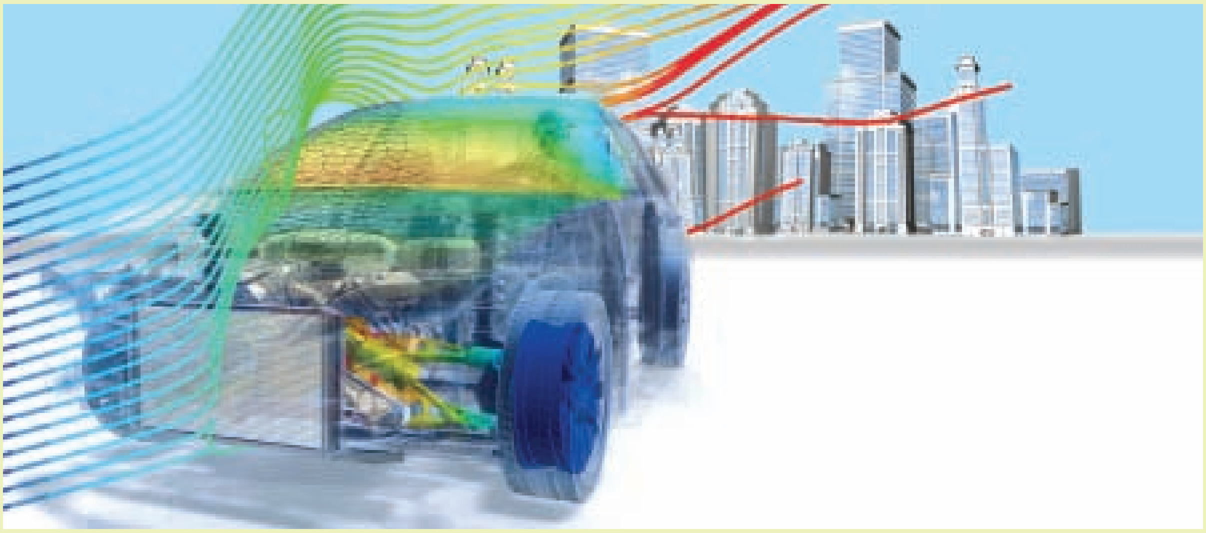
The ANSYS Workbench framework is designed to provide common services for engineering simulation applications — data management, parameterization, scripting and graphics, among others. Release 12.0 relies heavily on the framework's data management and parameterization services to integrate existing applications into the ANSYS Workbench environment, where they have become highly interoperable. Over subsequent releases, these applications will leverage the framework's graphical toolkit to establish a consistent user interface and further blend the various applications integrated into the platform. At the onset of developing ANSYS Workbench 2.0, we identified scripting and journaling as fundamental requirements of the new architecture. As such, a top-level scripting engine has been thoughtfully designed and lays the groundwork for future ANSYS Workbench customization and batch processing. Looking beyond release 12.0, all these services will be further refined and will fuel rapid add-in development and a further expansion of capabilities. Over time, ANSYS customers and partners will leverage the framework's open architecture, enlisting its services to create tailored applications, and will elevate ANSYS Workbench as an application development platform for the engineering simulation community.

Simulation Process and Data Management

ANSYS Workbench 2.0 is an environment in which a single analyst creates and executes one or more steps of an engineering simulation workflow. ANSYS Engineering Knowledge Manager (EKM) extends ANSYS Workbench by providing the tools to manage the work of a group of analysts and myriad simulation workflows. This includes system-level services to manage and foster collaboration on thousands of models, terabytes of results, hundreds of defined processes and huge investments in simulation.

Looking forward, ANSYS believes that managing data and processes will become integral with engineering simulation. Ten years ago, simulation comprised three discrete and sequential phases: pre-processing, solving and post-processing. With the evolution of ANSYS Workbench, we now look at engineering simulation as a continuous workflow intertwining these steps. In the same way, process and data management will become intertwined





As mechanical and electrical engineering worlds converge, the combination of ANSYS and Ansoft technologies will allow engineers to analyze the behavior of combined systems.

with simulation, expanding its role and aligning it with business processes such as product lifecycle and supply chain management.

Electromechanical System Simulation

The ANSYS acquisition of Ansoft anticipates a trend in the realm of engineering and design: The mechanical, electrical and software engineering worlds will rapidly converge. Several years ago, the synchronization of these worlds was coined "mechatronics," and, today, the combined disciplines are responsible for engineering the electromechanical systems found in everything from washing machines to airplanes. A simple examination of the automotive industry reveals that the more recent and exciting advancements have relied on mechatronics. So, at a time when greeting cards and tennis shoes contain microprocessors and sensors, mechatronics is not just for high-end cars and appliances; rather it is the key to unleashing innovation in every industry.

For many years, electrical and mechanical engineering teams have increasingly relied on simulation to accelerate innovation, but each camp has adopted simulation tools that were not fully capable of addressing the needs of the other — until now. As the separation between the electronic and mechanical worlds becomes increasingly blurred, ANSYS has extended its range of simulation technology by incorporating Ansoft's world-class product portfolio. Standardizing on ANSYS Workbench for Simulation Driven Product Development means establishing a common platform on which to further develop both mechanical and electronic components and analyze the behavior of the combined systems. Driving innovation with mechatronics will require a comprehensive electromechanical simulation environment developed by a leader in both mechanical and electronic simulation software.

The Future Begins Now

With its advancements in individual physics, high-performance computing, multidomain simulation, meshing, and key enabling technologies such as simulation workflow and data management, release 12.0 clearly delivers on the ANSYS vision for Simulation Driven Product Development. But even though we have come a long way with the advent of ANSYS 12.0, there is still an exciting journey ahead. Standing on the strong foundation of all that ANSYS has learned and developed in almost 40 years of leadership in engineering simulation, we see many new opportunities on the horizon that will extend the reach of how customers use our technology. The ANSYS vision and strategy continue to set our bearings, and we continue to invest in pioneering new frontiers of the industry. And most important is that we remain committed to enabling customers to use simulation to develop innovative products that perform better, cost less and are brought to market faster. ■

This article was written through contributions from Todd McDevitt of ANSYS, Inc.