Multiphysics for the Real World

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

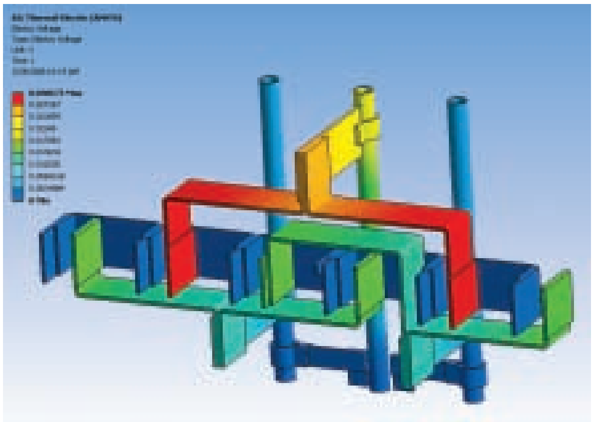
Continuing to build on the foundation of prior releases, ANSYS 12.0 expands the company's industry-leading comprehensive multiphysics solutions. New features and enhancements are available for solving both direct and sequentially coupled multiphysics problems, and the ANSYS Workbench framework makes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

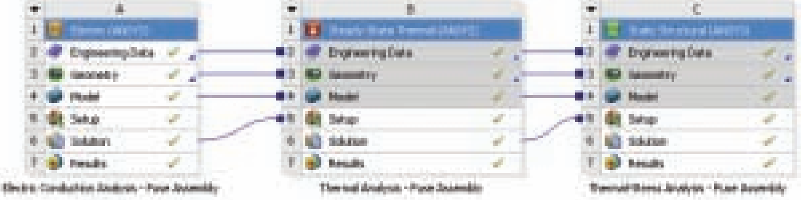
Also new in ANSYS Workbench at version 12.0 is support for direct coupled-field analysis. Relevant elements (SOLID226 and SOLID227) are now natively supported in the ANSYS Workbench platform for thermal–electric coupling. There also is a new analysis system for thermal–electric coupling that supports 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

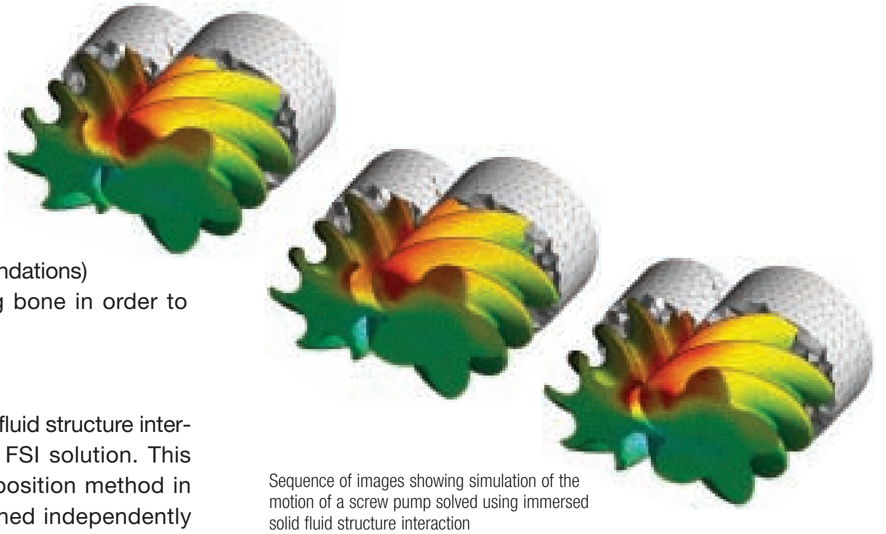
ANSYS 12.0 extends the distributed sparse solver to support unsymmetric and complex matrices for both shared and distributed memory parallel environments. This new solver technology dramatically reduces the time needed to perform certain direct coupled solutions including Peltier and Seebeck effects 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

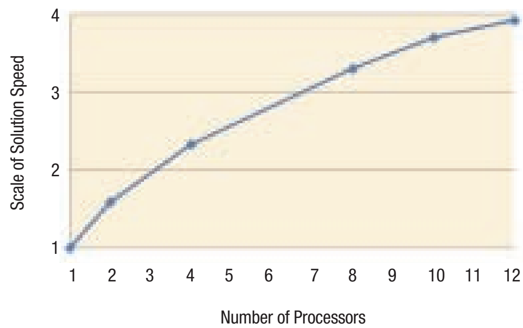
A new family of direct coupled-field elements is available in ANSYS 12.0; these new elements enable the modeling of fluid flow through a porous media. This exciting new capability, comprising coupled pore–pressure mechanical solids, enables multiphysics modeling of new classes of civil and biomedical 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.

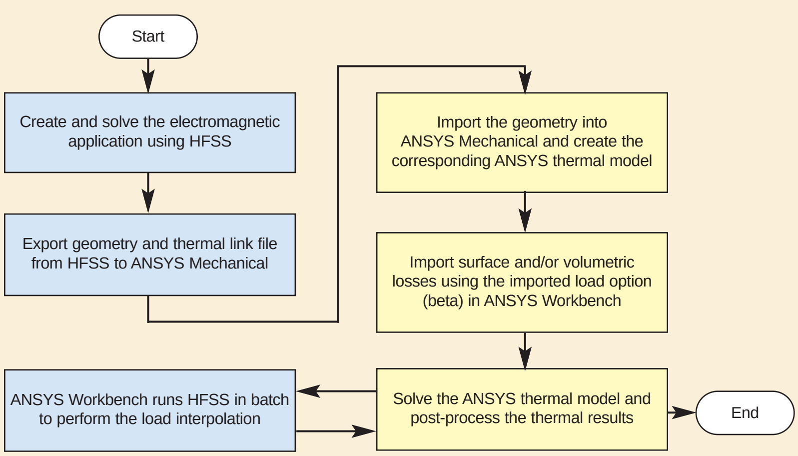


Solution scaling of a thermoelectric cooler model with 500,000 degrees of freedom enables a speedup of four times for 12 processors.

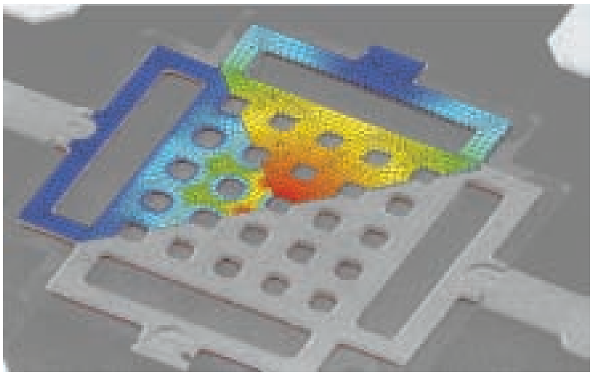
Coupling Electromagnetics

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

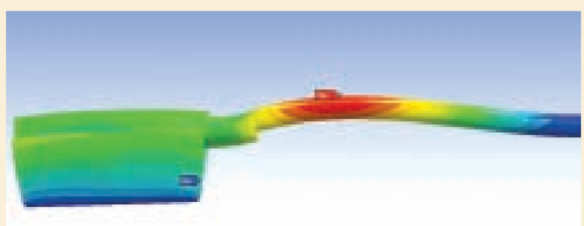
The multi-field solver (used for performing implicit sequential coupling) contains a number of new enhancements at release 12.0. The first is a new solution option that controls writing a multiframe restart file. This capability allows a user to restart an analysis from any multi-field time step, which 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.