

## **ANSYS LATEST RELEASE FEATURES INTEGRATED FLEXIBILITY AND PERFORMANCE**

***ANSYS 10.0 includes performance upgrade and superior coupled physics technology in fluid structure interaction***

**SOUTHPOINTE, PA – June 02, 2005** – ANSYS, Inc. (NASDAQ: ANSS), a global innovator of simulation software and technologies designed to optimize product development processes, today announced Version 10.0 of its ANSYS® software. The new release offers major advances in performance, ease-of-use, interoperability and coupled physics technology, such as fluid structure interaction (FSI). With availability beginning in July, this latest release builds upon and is completely compatible with the current ANSYS 9.0 software.

Continuing to improve upon its Multiphysics technology, ANSYS 10.0 offers superior capabilities for complex FSI problems. Representing a complete FSI solution from a single vendor, it utilizes each of the best-in-class technologies in stress and fluid flow analysis. A single geometry is used for both, with meshing appropriate to the specific physics required. A high-speed protocol provides the communication necessary to perform dynamic FSI analysis. No other FSI offering in the market today offers this degree of robustness and fidelity. Furthermore, it parallelizes on multi-machine clusters for very large problem sizes.

“ANSYS 10.0 represents the latest in integrated CAE capability, building upon the significant advances we delivered in ANSYS 9.0,” said Jim Cashman, president and CEO at ANSYS, Inc. “We have continued to increase the breadth and depth of the ANSYS simulation technology, while at the same time establishing an unprecedented level of interoperability between all forms of simulation and analysis. Taking full advantage of the ANSYS® Workbench™ infrastructure for integrated CAE, we have created a more seamless approach for meshing and modeling, simulation and analysis, and post-processing. Our world-class capabilities in each of the forms of analysis such as structural, thermal and fluids are all together as part of ANSYS 10.0.”

Further strengthening the industry-specific capabilities in the Workbench environment, ANSYS introduces design tools for rotating machinery and blade design. ANSYS 10.0 features ANSYS® BladeModeler™, an efficient 3-D rotating machinery design tool for bladed components and ANSYS® TurboGrid™, a high quality hexahedral meshing tool for blade design.

“Combined with ANSYS® CFX® and specialized turbo pre- and post-processing CFD capabilities, this creates a comprehensive solution set for turbomachinery design and analysis,” said Chris Reid, vice president and general manager at ANSYS, Inc. “Models for stress analysis, computational fluid

dynamics or fluid structure interaction can be created, expanded to include upstream and downstream components via CAD system connectivity, and then fully analyzed. ANSYS® Workbench™ is a unique environment to provide for this, such that the aerodynamics engineer can perform CFD-intensive design, while also confirming the structural characteristics. This offers the potential for significant reduction in design cycle times.”

In the mechanical application area, ANSYS 10.0 includes thermal transients, fully implemented within ANSYS Workbench. This enables users to perform very sophisticated time-based simulations while the ANSYS Workbench integration automates many of the model setup and solver tasks. The result is a fast and easy-to-use solution for studying a product’s thermal performance over the expected operating time period.

To address the broader need for efficient and timely execution of analysis problems of ever-increasing size and complexity, the ANSYS 10.0 parallel solver now supports new choices in both communications technology as well as processors. In addition to Ethernet and Gigabit Ethernet (GigE), ANSYS 10.0 supports Myrinet and InfiniBand. This allows for clusters of very high performance at significantly less cost than previous configurations.

Furthering our solutions on high-performance and low-cost hardware configurations, ANSYS Workbench now is supported on Windows XP 64-bit for AMD and EMT64 chipsets. This support eliminates the 2GB memory limit that many users face when running large models on Windows. In addition, it provides the opportunity to add extra physical memory which ANSYS uses to complete the entire solution without writing to the hard disk – usually the source of extended computation times.

For customers, this enables large model sizes, such as those found in low-frequency static and full transient electromagnetic analyses, to be addressed more efficiently and economically. Plus, the ANSYS 10.0 parallel solver now supports these physics, making ANSYS the leading CAE vendor able to address large electromagnetic problem sizes that are greater than 100 million degrees of freedom.

In the area of high-frequency electromagnetics a new modal port definition is available. This port enhancement greatly simplifies the model setup for transmission line ports needed for many types of Integrated Circuit (IC), Radio Frequency IDentification (RFID) and Radio Frequency (RF) MicroElectroMechanical Systems (MEMS) device analysis. Benchmarks show typical reduction in model size resulting in a 30 to 50 percent reduction in solution times and memory requirements for a given model while still providing more accurate frequency-dependent results.

Other developments in this version include the addition of gyroscopic effects that enhance the performance of ANSYS for rotor dynamics application of turbomachinery and other rotating structures. In the area of direct-coupled field physics, structural-thermal-electric coupling has been enhanced to

include thermoelastic damping (TED), an important internal loss mechanism in metals, ceramics and MEMS (resonator beams).

ANSYS continues to deliver powerful capabilities that enable our customers their choice of full automation as well as individual control against our ANSYS Workbench theme. We have made significant enhancements to core meshing architecture to share meshes across ANSYS Workbench applications and improvements in CAD robustness. ANSYS® ICEM CFD™ 10.0 continues to take meshing to a new level by providing a complete set of tools to model real world systems, like under-hood and crash analysis, through new capabilities related to hybrid meshing as well as handling of detailed CAD models.

“ANSYS 10.0 represents another significant step forward for ANSYS, within each of the physics domains, and in the way they can be utilized together in the ANSYS Workbench,” said Mike Wheeler, vice president and general manager at ANSYS, Inc. “No other company in the world of CAE can match the comprehensiveness of our solution set or provide the same level of integration in a single CAE environment.”