



## **ANSYS Latest Release Features Unique Simulation Technologies and Integrated Solutions**

### **ANSYS 11.0's Powerful Enhancements Increase Productivity and Broaden Engineering Simulation**

[Visit News Releases Image Library](#)

SOUTHPOINTE, Pa., Feb. 26 /PRNewswire-FirstCall/ -- ANSYS, Inc. (Nasdaq: ANSS), a global innovator of simulation software and technologies designed to optimize product development processes, today announced Version 11.0 of its ANSYS® software. This latest version of the ANSYS family of engineering simulation solutions offers new and enhanced tools and capabilities that enable users to complete jobs efficiently and fully leverage Simulation Driven Product Development for a wide range of applications. This release represents the leading edge in integrated, best-in-class computer-aided engineering (CAE) functionality including advanced analysis, meshing, optimization, multiphysics and multibody dynamics.

The enhancements to ANSYS 11.0 software follow the tradition of making powerful simulation tools available to an increasingly wider range of product developers. "As with previous major product releases, we are pleased that ANSYS continues to deliver according to customer expectations: better and more expansive tools, all in pursuit of true Simulation Driven Product Development," says Jim Cashman, president and CEO of ANSYS, Inc.

Continuing its commitment to provide state-of-the-art solutions, ANSYS introduces new solver technology designed to reduce time when performing transient solutions. Version 11.0 uses Variational Technology (VT) as an advanced predictor-corrector algorithm to reduce the overall number of iterations for nonlinear static and transient analyses. In the CAE field, this kind of analyses traditionally results in long run times - and has discouraged users from performing what-if scenarios that might provide information about how a design performs in the real-world environment. By reducing the overall number of iterations, ANSYS has greatly reduced the solution time for nonlinear static and transient analyses. This capability provides a 2X to 5X speedup for the initial solutions, depending on the hardware, model and type of analysis used. ANSYS® VT Accelerator™ technology makes re-solves 3X to 10X faster for parameter changes, allowing for effective simulation-driven parametric studies of nonlinear and transient analyses in a cost-effective manner.

In order to provide "innovations that work," Florida Turbine Technologies, Inc. - which executes all aspects of turbine engine design and development in the military and commercial aircraft industry - desires transient fidelity early in the design process. "Due to long run times, we usually reserve transient analyses for detailed final design," says Joseph T. Metrisin, lead structures engineer at Florida Turbine Technologies, Inc. "Faster solution options will allow us to perform detailed transient analyses early on in the design process, resulting in more robust designs."

The integration of ANSYS and ANSYS® CFX® technologies in the ANSYS® Workbench™ platform has taken another step forward. With version 11.0, users will be able to set up, solve and post-process a two-way fluid structure interaction (FSI) simulation completely in ANSYS Workbench. The latest release also provides a single post-processing tool. ANSYS Workbench significantly reduces the time needed to obtain solutions to complex multiphysics phenomena.

ANSYS 11.0 has been refined to address some industry-specific needs as well. In the rotating machinery sector, for example, ANSYS leverages the power of its advanced fluids (ANSYS CFX) and multiphysics (ANSYS) software, along with its integration platform (ANSYS Workbench), and combines these with a partner tool (Vista CC Design) that allows users to develop better turbomachines in shorter time. The ANSYS Workbench platform provides an integrated geometry design and analysis system that links all elements of the design process. It is the integration platform for advanced physics capabilities in ANSYS CFX and ANSYS® BladeModeler™ that enable designers to model rotating machinery such as pumps, compressors, fans, blowers, turbines, expanders, turbochargers and inducers. The integration of all these ANSYS solutions into the design process can take weeks out of the CAE process by eliminating manual file transfer, result translation and re-analysis time. The partner tool, initial sizing software used during the preliminary design phase, is from PCA Engineers Limited.

The first step in the turbomachinery design process is to obtain a preliminary design using initial sizing software, given the performance criteria and sizing constraints. At ANSYS 11.0, PCA Engineers Limited's Vista CC Design (CCD) initial sizing software for centrifugal compressors and pumps is included in ANSYS BladeModeler. This is a rapid meanline design program that - when given the compressor duty mass flow, pressure ratio and geometric constraints - configures the compressor scantlings, vane inlet and exit angles, velocity triangles. It also provides essential non-dimensional performance parameters, such as specific speed and specific flow rate on which design decisions can be based.

Turbo Solutions Engineering LLC has experience with a broad range of CAE software. It uses ANSYS solutions to help customers in industries such as aerospace and HVAC to develop new machines. "The recent addition of PCA Engineers' Vista CCD and Vista CPD compressor and pump design software into the ANSYS CFX product line has considerably streamlined our turbomachinery design process," says Nicholas D'Orsi, partner and chief engineer at Turbo Solutions Engineering. By using the Vista interface, we can rapidly complete iterative sizing and performance estimation for centrifugal and mixed-flow turbomachinery. A key feature of this new interface is the transfer of an initial impeller three-dimensional design into ANSYS BladeModeler software. Once inside ANSYS BladeModeler, the analysis features of ANSYS CFX allow for quick optimization of the design."

Integrating meshing technologies and applications to provide the widest selection of meshing tools has been a major focus for ANSYS in the past five years, and the latest release is no exception. ANSYS 11.0 delivers more examples of meshing technology integration and provides physics-based meshing solutions that tailor the mesh for mechanical, electromagnetics, computational fluid dynamics (CFD) or explicit dynamics simulation. Best-in-class meshing technology from ANSYS, ANSYS ICM CFD™ and ANSYS CFX products has been integrated in the ANSYS Workbench platform to leverage the strengths of the various algorithms to provide an intelligent, flexible and robust solution to meshing.

Based on the defined physics filter, various controls are automatically defined, such as mesh size, mesh transition, mesh uniformity, mesh speed, mesh quality and refinement controls for proximity and curvature. Advanced user controls then are available to exert influence over the mesh when required. This intelligence in meshing allows even the novice user to get a good mesh suited for the defined physics while providing the flexibility of additional controls to improve the solution speed and/or accuracy. The multiple meshing methods, available through advanced options, also provide backup meshing approaches to improve the overall robustness of the meshing solution.

Integration efforts now have been applied in the explicit dynamics arena as well, resulting in a modeling environment for explicit analysis that is much easier to use and more productive. ANSYS AUTODYN® software is a uniquely versatile explicit analysis tool for modeling the nonlinear dynamics of solids, fluids and gases and their interactions. At release 11.0, ANSYS AUTODYN is available for the first time as an integrated tool in the ANSYS Workbench platform. Tightly coupling ANSYS AUTODYN with tools such as ANSYS Meshing and ANSYS® DesignModeler™ provides an environment in which rapid decisions can be made based on results provided only by an explicit dynamics simulation.

A leading developer and manufacturer of ammunition systems as well as missile and space propulsion products, Nammo Raufoss AS uses advanced CAE tools to design and optimize new products. "At Nammo Raufoss AS, we have found that our preferred explicit dynamics tool, ANSYS AUTODYN, is even more useful to us now that it is part of ANSYS Workbench at release 11.0," says Gard Odegaardstuen, research and development manager at Nammo Raufoss. "We are saving significant amounts of time by being able to associatively link to our CAD models, mesh and solve them all within the ANSYS Workbench platform. Because we are able to parametrically modify the CAD and re-run a new simulation with a few mouse clicks, we are now able to use the ANSYS AUTODYN results to guide our design efforts."

Another significant new enhancement to legacy ANSYS technology allows CAE users to dramatically cut time spent studying design performance on flexible dynamic systems. At release 11.0, ANSYS enhances and assembles its wide array of advanced structural dynamics capabilities, bringing frequency response and time history of flexible structures and dynamic systems together. Users now can select a range of behaviors: from linear to advanced nonlinear fully flexible responses, and all combinations in between, for their design performance simulations.

At Dale Earnhardt, Inc., engineers use ANSYS tools to analyze racecar parts and performance for increased power, speed and safety. "In NASCAR, we are always looking for an advantage that produces a faster and safer race car," says John Klausmeier at Dale Earnhardt, Inc. "The enhanced flexible dynamics capabilities we've found in ANSYS 11.0 are helping us understand what happens to our machines at 200 miles per hour under real transient track conditions. These enhanced ANSYS tools show us not only where some components have failed previously, but also why they've failed. Because ANSYS tools maintain associativity with our CAD vehicle models, fixing the problem is pretty straightforward once we've discovered what is causing them to fail."

The goal of the ANSYS focused software development road map is to provide customers with the most advanced and reliable engineering simulation solutions available in the industry. Chris Reid, vice president, marketing at ANSYS, Inc., says, "With ANSYS 11.0, we have continued our track record of adding significant new and enhanced capabilities to address increasing complexity and sophistication - simulation requirements that come from our valued customers as well as the industry in general."

For downloadable, high-resolution images, visit [www.ansys.com/newsimages](http://www.ansys.com/newsimages).

About ANSYS, Inc.

ANSYS, Inc., founded in 1970, develops and globally markets engineering simulation software and technologies widely used by

engineers and designers across a broad spectrum of industries. The Company focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-conscious product development, from design concept to final-stage testing and validation. The Company and its global network of channel partners provide sales, support and training for customers. Headquartered in Canonsburg, Pennsylvania, U.S.A., with more than 40 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ approximately 1,400 people and distribute ANSYS products through a network of channel partners in over 40 countries. Visit [www.ansys.com](http://www.ansys.com) for more information.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners.

SOURCE ANSYS, Inc.

CONTACT: For media, Kelly Wall, +1-724-514-3076, [kelly.wall@ansys.com](mailto:kelly.wall@ansys.com), or investors, Lisa O'Connor, +1-724-514-1782, [lisa.oconnor@ansys.com](mailto:lisa.oconnor@ansys.com), both of ANSYS, Inc.