



ANSYS Launches Immersed Boundary Module for Rapid Design Evaluation

Automated Meshing Approach for Complex Fluid Flow Analysis Facilitates Simulation Driven Product DevelopmentTM

SOUTHPOINTE, Pa., Aug 31, 2009 (BUSINESS WIRE) -- ANSYS, Inc. (NASDAQ: ANSS), a global innovator of simulation software and technologies designed to optimize product development processes, today announced the availability of the Immersed Boundary module for ANSYS^(R) FLUENT^(R) 12.0 software. Jointly developed by ANSYS and long-time partner Cascade Technologies Inc., the module dramatically reduces the amount of time needed for fluid flow simulations and provides fast results by directly addressing challenges associated with meshing. This first release, which is fully parallelized, supports the physical models and boundary conditions needed for modeling low-speed external aerodynamics and automotive front-end airflows. The Immersed Boundary tool offers the ability to save customers time and money by reducing the human effort needed to go from computer-aided design (CAD) to an analysis solution.

A conventional fluid dynamics simulation starts with the transfer of CAD data to a grid-generation package, in which a surface mesh and then a volume mesh are generated before the simulation can be set up and the solution run. The effort and time required for such pre-processing tasks can be significant. For example, in cases with complex or dirty geometry that require CAD cleanup, this part of the process may take 50 percent to 90 percent of the total time required for the simulation. The Immersed Boundary module addresses such issues by providing a rapid, automated, preliminary design approach. "This is an ideal tool for reducing lead time during the early stages of the product development cycle, when it is crucial that design evaluations are made rapidly. The Immersed Boundary module helps engineers identify the most promising designs, then use traditional methods for the final high-fidelity analysis," said Dipankar Choudhury, vice president of corporate product strategy and planning at ANSYS, Inc. "Because of our software's open and flexible architecture, ANSYS can readily partner with others in the engineering simulation ecosystem to deliver immediate customer benefits -- in this case, we partnered with Cascade Technologies, which has many years of fluid dynamics experience, to bring about a more efficient design process." The immersed boundary method is fully compatible with high-performance computing offerings from ANSYS, which can further reduce analysis time.

Fluid flow simulations using the Immersed Boundary module for ANSYS FLUENT 12.0 software start with the surface data of the simulation geometry in the STL file format, which is commonly used in rapid prototyping and computer-aided manufacturing. This CAD geometry does not need to be clean, does not require smooth surfaces or geometry connectivity, and may contain overlapping surfaces, small holes and even missing parts. The simulation geometry is meshed automatically. Mesh refinement also is carried out automatically after specifying the desired resolution on the boundaries, ensuring the accuracy required for preliminary design evaluation. Using the immersed boundary meshing technique greatly reduces the amount of time spent preparing the geometry for meshing and creating the mesh.

About Cascade Technologies Inc.

Cascade Technologies Inc., located in Mountain View, California, U.S.A., develops, markets, and supports state-of-the-art computational fluid dynamics analysis tools for engineers across industries. A long-time ANSYS partner, the company developed innovative add-ons for ANSYS FLUENT software, including the V2F turbulence model and the immersed boundary modeling method. The company accelerates technology transfer to clients through the development of advanced computational modules specially designed for commercial CFD packages. Hence, cutting-edge developments in computational engineering are provided to the end-users in their preferred environment.

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 60 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ over 1,600 people and distribute ANSYS products through a network of channel partners in over 40 countries. Visit www.ansys.com for more information.

ANSYS, ANSYS Workbench, Ansoft, 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. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Technology: ANSS-T

SOURCE: ANSYS, Inc.

ANSYS, Inc.

Media:

Kelly Wall, 724-514-3076

kelly.wall@ansys.com

or

Investors:

Annette Arribas, 724-514-1782

annette.arribas@ansys.com

Copyright Business Wire 2009