Fluids Simulation Advancements In ANSYS 15.0 Bring Faster and More Accurate Results to Users

December 20, 2013

PITTSBURGH, Dec. 20, 2013 /PRNewswire/ -- With faster pre-processing, solver and optimization technologies, users of ANSYS Inc.'s (NASDAQ: ANSS) release 15.0 for fluid dynamics simulation will be able to meet tighter development deadlines and further reduce time to market.

(Logo: http://photos.prnewswire.com/prnh/20130430/NE03388LOGO)

Released earlier this month, new functionalities in ANSYS® 15.0 offer users a wealth of benefits across its product portfolio. As the second in a series of announcements highlighting improved capabilities across three main physics areas, ANSYS today focuses on the enhancements to its computational fluid dynamics (CFD) offerings at release 15.0. CFD solutions from ANSYS provide the ability to confidently predict that products impacted by flow phenomena will thrive in the real world. As products grow increasingly complex, the geometry and physics being simulated are also becoming more complicated. The advances in ANSYS 15.0's CFD solutions make it easier for users to arrive at fast, accurate solutions.

Thanks to smart optimization technologies, faster solver speed, ground-breaking parallel scalability, intuitive geometry functionality and parallel meshing technologies, users of ANSYS 15.0 for fluid dynamics will be able to design new products faster and more accurately than ever before.

Release highlights for computational fluid dynamics in ANSYS 15.0 include:

- Optimization is faster thanks to an improved adjoint solver. The adjoint solver now supports larger problems with meshes up to 30 million cells. The core functionality of the adjoint energy equation has been implemented in a way that observables can be defined as various integrals of heat flux and temperature — including averages and variances. As a result, the adjoint solver can now be used in problems where heat exchange maximization or heat loss reduction is a key design goal.
- ANSYS 15.0 provides numerous solver speed improvements. For example, simulations of immiscible fluid using the volume-of-fluid model are up to 36 percent faster. Transient Eulerian multiphase flow simulations are also accelerated thanks to adaptive time-stepping support. Dynamic combustion mechanism reduction can lead to simulation up to seven times faster with large chemical mechanisms.
- Simulation of fluid flow with heat transfer has been enhanced in numerous ways in ANSYS 15.0 multi-layer shell conduction greatly simplifies thermal management simulations and speeds up the overall workflow, avoiding the need to volume-mesh very thin material surfaces. Anisotropic thermal conductivity behavior for solid materials can also now be modeled. The surface-to-surface radiation model supports non-conformal meshes. This provides more flexibility in meshing both large fluid volumes together with thin fabricated structures, also helping to speed up the time to solution.
- Continuous improvements in ANSYS 15.0 with high-performance computing (HPC) scalability and robustness allow engineers to increase simulation throughput. One set of benchmark results demonstrated scalability above 80 percent efficiency with as few as 10,000 cells per compute core, representing a three-fold improvement over previous releases. ANSYS 15.0 also reduces the time needed to read a simulation file and start the simulation on large HPC clusters, in some extreme cases going from 30 minutes to 30 seconds. Furthermore, a benchmark example of a flow simulation with large number of particles showed 80 percent parallel efficiency on 6,000 computing cores with ANSYS Fluent®. Thanks to efforts to improve ANSYS CFX® scalability on larger core counts, turbomachinery and other applications are being sped up significantly. Also, GPUs can be used with the AMG coupled pressure-based solver in ANSYS Fluent, with benchmarks here showing a speed up of up to 2.5 times is possible.

  > "Thanks to ANSYS fluid dynamics simulation solutions' speed and unmatched scalability on high-performance computer clusters, we are able to quickly and accurately assess the performance of a large number of designs," said Rob Rowsell of Wirth Research.

- Multiphysics simulations are easier to setup and run thanks to the introduction of 2-way thermal and 2-way thermal with force/displacement with ANSYS Fluent and ANSYS Mechanical via system coupling. This solution fully automates simulations which were previously extremely time consuming to setup and perform.
- Pre-processing complex CAD geometry is much faster, thanks to the introduction of advanced diagnostic and repair tools that guide the user when dealing with problematic geometries. Geometry problems like holes and gaps are automatically detected and can be quickly remedied with limited manual intervention. Major speed and memory use improvements have been implemented for all meshing methods. In addition, parallel meshing delivers dramatic reductions in mesh generation time. For example, for a 42 million-cell mesh, meshing time is reduced by up to 7.4 times when multiple processors are used.

ANSYS 15.0 contains many additional capabilities, which are detailed at http://www.ansys.com/ansys15.0.

Visit our blog or the ANSYS 15.0 webinar series for more information.

Current customers can download the latest version on the ANSYS Customer Portal.

About ANSYS, Inc.
ANSYS brings clarity and insight to customers' most complex design challenges through fast, accurate and reliable engineering simulation. Our technology enables organizations — no matter their industry — to predict with confidence that their products will thrive in the real world. Customers trust our software to help ensure product integrity and drive business success through innovation. Founded in 1970, ANSYS employs more than 2,500 professionals, many of them expert in engineering fields such as finite element analysis, computational fluid dynamics, electronics and electromagnetics, and design optimization. Headquartered south of Pittsburgh, U.S.A., ANSYS has more than 75 strategic sales locations throughout the world with a network of channel partners in 40+ countries. Visit www.ansys.com for more information.

ANSYS also has a strong presence on the major social channels. To join the simulation conversation, please visit: www.ansys.com/Social@ANSYS

ANSS-T

Media:
Jackie Mavin
724.514.3053
Jackie.mavin@ansys.com

Investors:
Annette Arribas, CTP
724.514.1782
annette.arribas@ansys.com

SOURCE ANSYS, Inc.