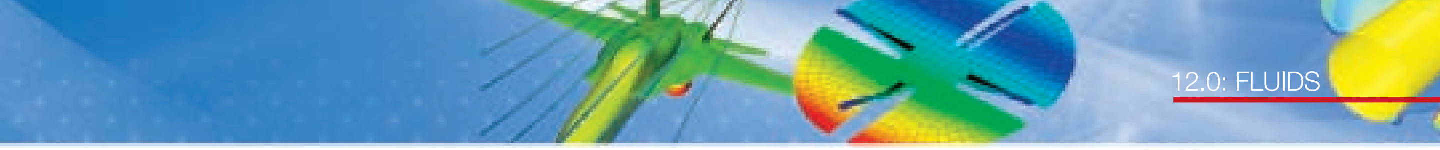
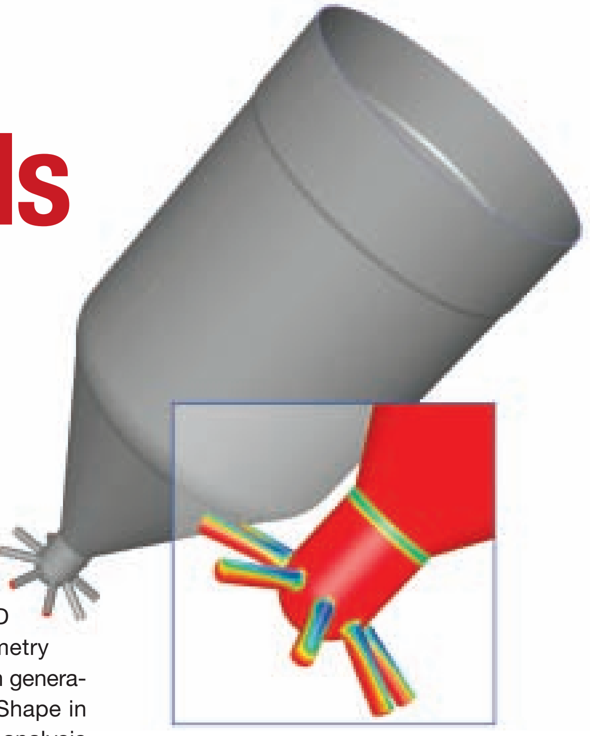


A Flood of **Fluids Developments**

A new integrated environment and technology enhancements make fluids simulation faster, more intuitive and more accurate.



Fuel injector model with close-up of vapor volume fraction contours at the injector surface

With release 12.0, ANSYS continues to deliver on its commitment to develop the world's most advanced fluid dynamics technology and make it easier and more efficient to use. Through its use, engineers can develop the most competitive products and manufacturing processes possible. In addition to delivering numerous new advancements in physics, numerics and performance, ANSYS has combined the functionality of both ANSYS CFX and ANSYS FLUENT into the ANSYS Workbench platform. Customers can use this integrated environment to leverage simulation technology, including superior CAD connectivity, geometry creation and repair, and advanced meshing, all engineered to improve simulation efficiency and compress the overall design and analysis cycle.

Integration into ANSYS Workbench

ANSYS 12.0 introduces the full integration of its fluids products into ANSYS Workbench together with the capability to manage simulation workflows within the environment. This allows users — whether they employ ANSYS CFX or ANSYS FLUENT software (or both) — to create, connect and re-use systems; perform automated parametric analyses; and seamlessly manage simulations using multiple physics all within one environment.

The integration of the core CFD products into the ANSYS Workbench environment also provides users with access to bidirectional CAD connections, powerful geometry modeling and advanced mesh generation. (See the article Taking Shape in 12.0.) Users can examine analysis results in full detail using CFD-Post, also available within the ANSYS Workbench environment.

Multiphysics

In some cases, fluid simulations must consider physics beyond basic fluid flow. Both ANSYS CFX and ANSYS FLUENT technologies provide many multiphysics simulation options and approaches, including coupling to ANSYS Mechanical software to analyze fluid structure interaction (FSI) within the ANSYS Workbench environment.

Another new capability is the immersed solid technique in ANSYS CFX 12.0 that allows users to include the effects of large solid motion in their analyses. (See the article Multiphysics for the Real World.)

General Solver Improvements

ANSYS continues to make progress on basic core solver speed, a benefit to all users for all types of applications, steady or transient. A suite of cases that span the range of industrial applications has consistently shown increases in solver speed of 10 to 20 percent, or even more, for both ANSYS CFX and ANSYS FLUENT software. Beyond core solver efficiency, improvements to various aspects of parallel efficiency address the continued growth and needs of high-performance computing. (See the article The Need for Speed.)

The perennial goal of improving accuracy without sacrificing robustness motivated numerous developments, including new discretization options such as the bounded second-order option in ANSYS FLUENT and the iteratively-bounded high-resolution discretization scheme in ANSYS CFX. Being able to consistently use higher-order discretization schemes means that users will see further increases in the accuracy of flow simulations without penalties in robustness.

User Interface

Ease of use has been enhanced in various ways. Most noticeably, the ANSYS FLUENT user interface has taken a significant step forward by adopting a single-window interface paradigm, consistent with other applications integrated in ANSYS Workbench. A new navigation pane and icon bar and new task pages and tools for graphics window management all reflect a more modern and intuitive interface while providing access to the previous version's menu bar and text user interface.

For ANSYS CFX software, a host of improvements have been added to the graphical user interface (GUI). There is a completely new capability that allows users to customize GUI appearance, including the option to create additional input panels. These custom panels provide the ability to encapsulate best practices and common processes by giving the user control over GUI layout and required input.

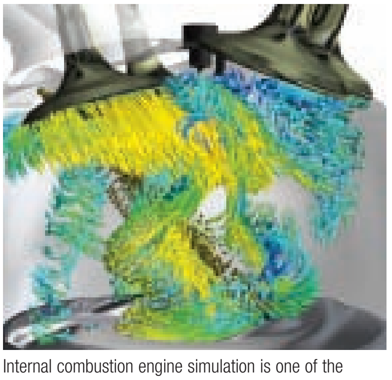
Evolution of the free surface of oil in a reciprocating compressor. The blue area is the gas/oil rotating domain inside the shaft, and the gray surface at the bottom shows the oil level of the reservoir. As the shaft rotates, oil is pumped up due to body forces. Image courtesy Embraco.

Specific Focus Areas

Internal Combustion Engines

Internal combustion (IC) engines are a primary target application for the development of numerous features. While this development is driven by the specific needs of IC engine simulations, it benefits many other applications and users:

- New options and flexibility for handling variations in physics complexity required at different phases of analyses
- Further-integrated options and controls for remeshing, including an IC-specific option for setting up an entire engine simulation
- Extensions and improvements to discrete particle-tracking capabilities
- Numerous enhancements to combustion models and their usability



Internal combustion engine simulation is one of the focus applications for ANSYS 12.0. This snapshot from a transient simulation of the complete engine cycle shows the flow just after the intake valves open and the direct injection of fuel. New flow feature extraction options in CFD-Post are used to highlight vortex structures with velocity vectors. Image courtesy BMW Group.

Multiphase

Multiphase flow modeling continues to receive a great deal of development attention, in terms of numerics and robustness improvements as well as extended modeling capabilities. ANSYS FLUENT software extends the single-phase coupling technology, introduced previously for the pressure-based solver, to include Eulerian multiphase simulations. This enhancement provides more robust convergence, especially for steady-state flows. ANSYS CFX users will find that improvements to the option to include solution of the volume fraction equations as part of the coupled set of equations make it more broadly usable in applications with separate velocity fields for each phase. Other modeling enhancements include the implementation of a wall boiling model and additional non-drag forces in ANSYS CFX as well as more robust cavitation and immiscible fluid models in ANSYS FLUENT.

Turbomachinery

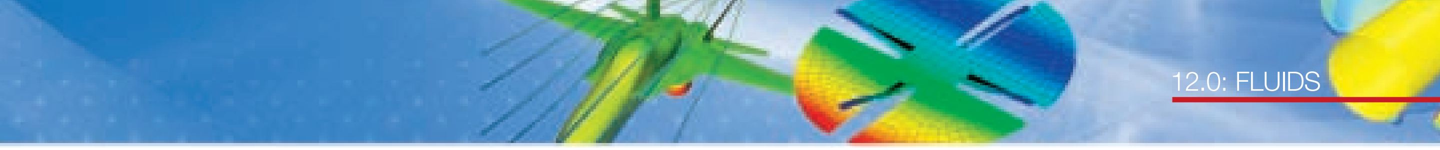
The significant proportion of customers using products from ANSYS for the design and optimization of rotating machinery ensured that this field received a substantial development focus. This latest release contains a variety of enhancements to core solver technology that couple rotating and stationary components more robustly, more accurately and more efficiently. ANSYS BladeModeler and ANSYS TurboGrid, specialized products for bladed geometry design and mesh generation, continue to evolve and improve. (See the Geometry and Meshing article for more details.)

An exciting new development for turbomachinery analysts is the introduction of the through-flow code ANSYS Vista™ TF. Developed together with partner PCA Engineers Limited, Vista TF complements full 3-D fluid dynamics analysis to provide basic performance predictions on one or more bladed components in a matter of seconds, allowing users to quickly and easily screen initial designs.

And More …

These enhancements represent just the tip of the iceberg in new and improved models and capabilities within core fluids products from ANSYS. Some other new developments include:

- Turbulence modeling extensions and improvements
  - Reynolds-averaged Navier–Stokes (RANS) models
  - Laminar–turbulent transition
  - Large eddy simulation (LES)
  - Detached eddy simulation (DES)
  - Scale-adaptive simulation (SAS)
- Ability to use real gas properties with the pressure-based solver in ANSYS FLUENT and, therefore, include these in reaction modeling
- Faster, more accurate chemistry across the board
- Dramatic speedups in view factor calculations in ANSYS FLUENT



"ANSYS CFX 12.0 showed a 30 percent solver speedup in comparison with the previous release. This significant improvement allows us to examine more design variations in the same time, enabling further design optimization and considerably reducing the total development time. This helps Embraco bring our products to the market more quickly."

— Celso Kenzo Takemori
Product and Process Technology Management
Embraco

- Inclusion of convective terms in solids to model conjugate heat transfer in moving solids in ANSYS CFX
- Ability to model thin surfaces in ANSYS CFX
- Much more in areas such as particle tracking, fuel cells, acoustics, material properties and population balance methods

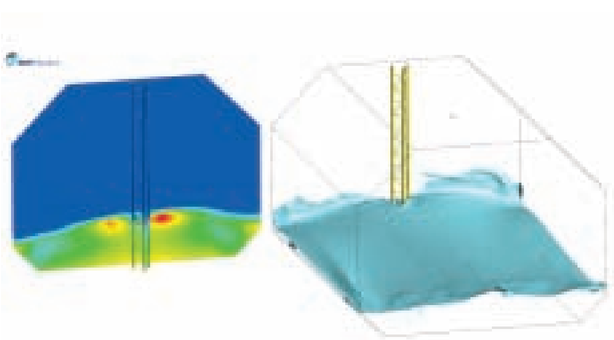
CFD-Post

An exciting introduction is the common post-processing application CFD-Post. The result of combining technologies from both ANSYS FLUENT and ANSYS CFX tools and building upon the well-established CFX-Post application, CFD-Post provides a complete range of graphical post-processing options to allow users to visualize and assess the flow predictions they have made and to create insightful 2-D and 3-D images and animations. The application includes powerful tools for quantitative analysis, such as a complete range of options for calculating weighted averages and automatic report-generation capabilities. All steps can be scripted, allowing for fully automated post-processing. Among the specific enhancements in release 12.0 are the ability to open and compare multiple cases in the same CFD-Post session and the addition of tools to locate vortex cores in the predicted flow field.

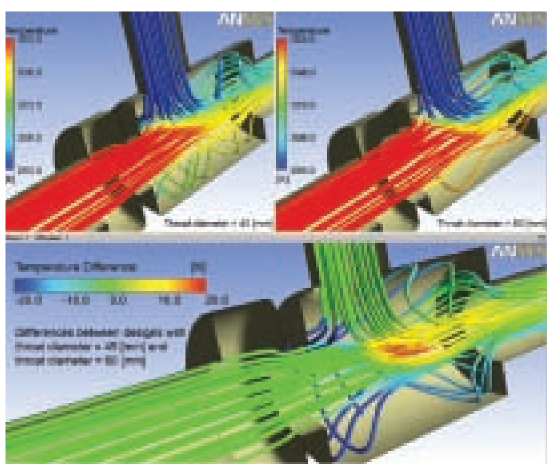
Conclusion

This is only a sampling of what the fluid dynamics development teams have produced for ANSYS 12.0. The combined depth and breadth of CFD knowledge and experience is delivering benefits to all users as technologies are combined and development teams drive simulation technology to new levels of achievement. With release 12.0, ANSYS continues its commitment to provide leading-edge CFD technology. ■

This article was written through contributions from Chris Wolfe and John Stokes of ANSYS, Inc.



In work sponsored by BMT Seatech, partially-filled tanks on marine vessels are being simulated by researchers at the University of Southampton to predict structural loads and changes in vessel behavior due to the sloshing of the fluid.



CFD-Post can be used to compare multiple designs directly, both by examining them side by side and by looking at the calculated difference between results.
Geometry courtesy CADFEM GmbH.