Cray and ANSYS Give Aircraft Takeoffs a Big Lift

For an airline, one of the most costly parts of its business is takeoff. That's because next to labor, airlines spend the most on jet fuel. And planes use the most fuel — and, consequently, emit the most harmful emissions — during takeoff.

An aircraft design component called a "high-lift device" can help make planes more fuel efficient. Broadly, a high-lift device is a movable surface (such as a flap) that increases lift during flight — most often in the takeoff and landing phases.

Effective high-lift devices can have a favorable influence on aircraft performance and operational costs. An improvement of even 1 percent in the lift-to-drag ratio could mean accommodating an additional ton of payload for a large aircraft.

Challenge

Understanding the reason for effective high-lift devices is simple. Modeling the air flow critical to their design is not. High-fidelity flow simulations of high-lift configurations require complex phenomena such as boundary-layer transition, flow separation, reattachment, and wake-boundary layer interaction. Additionally, the simulation must have sufficient resolution and the computational fluid dynamics (CFD) flow solver must have sufficient performance.

The simulation puts intense demands on compute system performance and requires robust and accurate software to model the complex physics.

> "This project demonstrates that complex high-lift aerodynamics can be accurately predicted with ANSYS Fluent software running on a Cray XC supercomputing system."
>
> *- Wim Slagter, Director of HPC and Cloud Alliances, ANSYS*

Cray and CFD software vendor ANSYS tackled the high-lift simulation challenge, working together on two high-lift prediction test cases developed by the AIAA CFD High Lift Prediction Workshop (HiLiftPW).

Organized by the AIAA Applied Aerodynamics Technical Committee, HiLiftPW assesses the numerical prediction capability of current-generation CFD technology/codes for swept, medium-to-high-aspect ratio wings for landing/takeoff (high-lift) configurations.

The HiLiftPW-3 test cases assessed two high-lift configurations:

- Grid convergence on the NASA High-Lift Common Research Model at four different grid refinement levels: coarse (~18 million cells), medium (~47 million cells), fine (~118 million cells), and extra fine (~397 million cells) with two angles of attack (8, 16 degrees)
- Nacelle and pylon installation on the JAXA Standard Model in nominal landing configuration with support brackets and nacelle pylon on/off at six different angles of attack (4.360, 10.470, 14.540, 18.580, 20.590, 21.570 degrees) using fine (~165 million cells) refinement level grid

Solution

Solving these demanding problems in a meaningful timeframe required a large number of compute cores. For example, one 165-million-cell simulation used 2,200 cores.

ANSYS® Fluent® is designed to take advantage of multiple cores using its MPI-based parallel implementation. Efficiently executing in parallel across a large number of cores depends on moving large amounts of data between the cores. In response, the Cray® XC™ system maximizes interprocessor communication performance, allowing the cores to spend their time computing.

One of the XC system's key features is the low-latency, high-bandwidth Aries™ interconnect. The Aries technology enables effective scaling for demanding simulations, regardless of other jobs running on the system.

In conjunction with the interconnect, the XC system uses a proprietary MPI library optimized to take advantage of the network's unique features. ANSYS Fluent is built with the Cray MPI library, which means it delivers exceptional parallel performance. In addition, the Cray Linux® Environment, Cray's high-performance software suite and operating system, has been optimized to reduce overhead, allowing more of the cores' cycles to be used for productive computation.

Results

The computational results made possible by Cray XC supercomputing power and obtained with ANSYS Fluent showed excellent performance.

For example, one 189-million-cell simulation completed in under two hours. Computational results were validated with experimental results from JAXA's Low Speed Wind Tunnel and showed excellent matching. Comparisons were based on aerodynamic forces, moments and pressure coefficients, along with surface oil flow visualizations.

CRAY XC SERIES

Cray XC series supercomputers handle the most challenging simulation, analytics and AI workloads with extreme scalability and sustained performance. See how they can help solve your biggest questions, too.

cray.com/products/computing/xc-series

ANSYS FLUENT

ANSYS Fluent is a CFD software solution used to predict flow, turbulence, heat transfer and reactions for industrial applications. Its advanced solver technology provides fast, accurate CFD results in the shortest possible time.