ANSYS® Fluent® Helps Reduce Vehicle Wind Noise

ANSYS Fluent

ANSYS Fluent is a computational fluid dynamics (CFD) software solution used to predict flow, turbulence, heat transfer and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from cleanroom design to wastewater treatment plants. Advanced solver technology provides fast, accurate CFD results, flexible moving and deforming meshes, and superior parallel scalability. Additionally, Fluent has a record of outstanding parallel scalability up to tens of thousands of CPU cores, enabling high-fidelity results in the shortest possible time.

Cray® XC40™ Supercomputer

The Cray XC40 supercomputer is a massively parallel processing architecture focused on producing more capable HPC systems to address a broad range of user communities. The system leverages the combined advantages of next-generation Aries interconnect and Dragonfly network topology, Intel® Xeon® processors, integrated storage solutions and major enhancements to the Cray OS and programming environment. The Cray® XC® series supercomputer is a groundbreaking architecture upgradable to 100 petaflops per system and delivering:

- Sustained and scalable application performance
- Tight HPC optimization and integration
- Production supercomputing
- Investment protection – upgradability by design
- User productivity

Situation

The drivetrain, the tires on the road and the aerodynamics: these are the factors that add noise to every driving experience. At low speed and high engine load, drivetrain noise dominates. At low speed and low engine load, it's where the rubber meets the road that most noise originates. Things change at higher speeds, where the aerodynamic noise of ground vehicles becomes dominant at driving speeds above 100 kph.

Interior noise isn't just an annoyance that makes it difficult for passengers to converse or enjoy their music. It can also be a hazard because it contributes to driver fatigue, particularly on long trips. To stay ahead of the competition and meet consumers' ever-increasing expectations for a comfortable ride, automotive OEMs are using ANSYS computational fluid dynamics (CFD) simulation software to tackle the problem of wind noise. Used early in the product development process, the software helps designers predict wind noise and then alter their designs to reduce it.

Challenge

When a vehicle – in this application, the Alfa Romeo Giulietta – is driven at high speed, the side-view mirrors are a major source of broadband (up to about 5,000 Hz) wind noise. The low-frequency range is dominated by large alternating vortices shed from the top and bottom edges of each mirror. This highly energetic wake, which resembles a Von Karman vortex street, strikes the side windows, creating noise.

Another source of wind noise at the side windows is the A-pillar vortex, which originates at the pillar between the windshield and the side window. A quantitative description of the wall pressure fluctuations on the side window requires knowledge of the fine details of the flow, which can be achieved only through accurate prediction of complex flow features, including time-dependent vortex shedding, large-to-small turbulent scales, convection and fluid compressibility. Detached-eddy simulation (DES) is a novel approach that combines the concepts of unsteady Reynolds-averaged Navier-Stokes (URANS) and large-eddy simulation (LES) to obtain realistic solutions for practical high-Reynolds-number flows at acceptable computational costs. Delayed detached-eddy simulation (DDES) is a version of DES designed to ensure that attached boundary layers are treated inside the RANS zone to avoid modelled stress depletion.

The DDES turbulence model was used to simulate side-window wall-pressure fluctuation noise for the Alfa Romeo Giulietta, and a very accurate transient simulation was performed using ANSYS Fluent CFD software running on a Cray® XC40™ supercomputer. To ensure accurate results, the test employed a very fine hexcore mesh of 62 million cells to capture both large- and small-scale fluctuation.

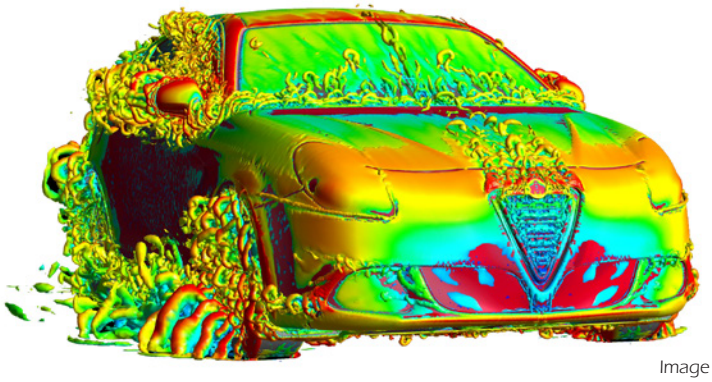

Image courtesy of FCA Italy

Cray Inc.
901 Fifth Avenue, Suite 1000
Seattle, WA 98164
Tel: 206.701.2000
Fax: 206.701.2500
www.cray.com

Solution

Solving this demanding problem in a meaningful time frame requires a large number of compute 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. The Cray XC40 system is designed to maximize the performance of interprocessor communication, allowing the cores to spend their time computing. One of the key features of the XC40 supercomputer is its proprietary Aries interconnect, which delivers low latency, high bandwidth and adaptive routing. This allows effective scaling for demanding simulations like this one, regardless of other jobs running on the system.

In conjunction with the Aries interconnect, the XC40 system uses a proprietary MPI library optimized to take advantage of the unique features of Aries. ANSYS Fluent is built with the Cray MPI library, which means it delivers exceptional parallel performance. In addition, the Cray Linux® Environment, Cray's Linux-based operating system, has been optimized to reduce overhead, allowing more of the cores' cycles to be used for productive computation.

To ensure that users of ANSYS Fluent and Cray's scalable systems benefit from the best possible performance, engineers from ANSYS and Cray closely collaborate to continually improve the performance of Fluent on Cray systems. Guided by mutual customers with demanding problems, the teams have identified numerous performance improvements, which ANSYS has incorporated into Fluent releases over the last four years.

The curve below shows the scaling performance of ANSYS Fluent 16.0 for the Alfa Romeo Giulietta aeroacoustics case.