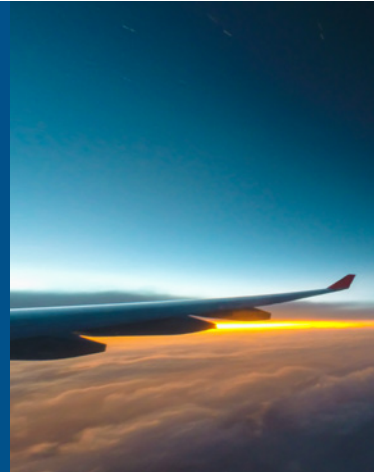# PREDICTING COMPLEX HIGH-FIDELITY AEROELASTIC WING FLUTTER

HIGH-FIDELITY SIMULATIONS OF THE AEROELASTIC FLUTTER EXPERIENCED BY AN AIRPLANE IN FLIGHT PUT EXTREME DEMANDS ON MACHINE PERFORMANCE. A COMBINATION OF CRAY HARDWARE AND ANSYS SOFTWARE SCALED A 13.47-MILLION-NODE SIMULATION TO MORE THAN 1,000 CORES.

## Challenge: Complex Aeroelastic Modeling

Airplanes are exposed to strong inertial and aerodynamic forces during flight, and the interaction of these forces with the airplane structure results in a complex phenomenon called aeroelasticity. Aeroelastic simulation is becoming increasingly important as aerospace structures become lighter and new materials and manufacturing techniques enable more innovative designs. An aeroeleastic failure can be catastrophic, and even for this relatively simple geometry the aeroelastic effects are significant.

Testing in this area is so critical that, to develop comprehensive benchmarking standards, an AIAA Aeroelastic Prediction Workshop (AePW) was established to assess state-of-the-art computational methods and tools to predict aeroelastic phenomena.

High-fidelity simulations of complex aeroelastic flutter phenomena are computationally intensive, putting extreme demands on overall system performance. Simulations across the full frequency range are required to ensure robust design — which requires quality software to model the complex physics with an HPC system that can handle the simulation accuracy and volume needed to explore the design space. The combination of Cray hardware and ANSYS software solves these demanding problems efficiently and delivers fast results.

> High-fidelity simulations of complex aeroelastic flutter phenomena are computationally intensive, putting extreme demands on system performance.

The results of this project demonstrate that complex aeroelastic wing flutter phenomena can be accurately predicted with ANSYS CFX software running on a Cray XC supercomputing system.

**THIS PROJECT IS JUST ONE EXAMPLE OF HOW ANSYS AND CRAY COLLABORATE TO BUILD ROBUST SOLUTIONS FOR A BROAD SET OF ENGINEERING SIMULATIONS.**
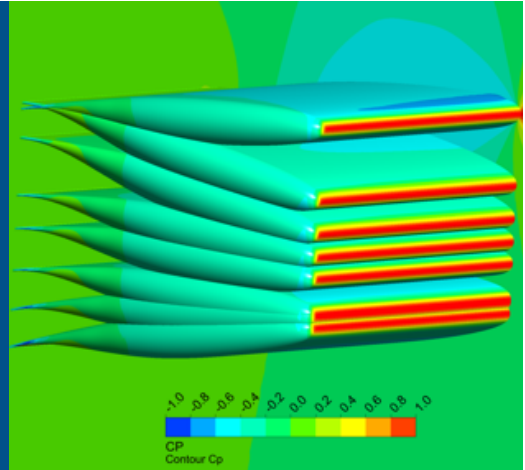
## About ANSYS® CFX®

ANSYS CFX software is a high-performance, general-purpose computational fluid dynamics (CFD) program that has been applied to solve a wide range of fluid flow problems for over 20 years. At the heart of ANSYS CFX is its advanced solver technology, the key to achieving reliable and accurate solutions quickly and robustly. The modern, highly parallelized solver enables ANSYS CFX to capture virtually any type of phenomena related to fluid flow in numerous physical models. The solver and its models are wrapped in a modern, intuitive and flexible GUI and user environment, with extensive capabilities for customization and automation using session files, scripting and a powerful expression language.

## About Cray® XC™ Series Supercomputers

Cray's XC system combines resiliency with leading edge performance, for proven increases in model fidelity and accuracy delivering greater product insight and reduced risk. Cray leverages the market-leading Aries™ interconnect and Dragonfly network topology, Intel® Xeon® processors, integrated storage solutions and enhanced Cray OS and programming environment.

Cray's groundbreaking architecture — upgradable to 500 petaflops per system — and unparalleled team expertise deliver:

- Sustained and scalable application performance
- Tight application optimization and integration
- Investment protection — upgradability by design
- Enhanced user productivity

## Cray XC Supercomputer + ANSYS CFX

ANSYS CFX scales efficiently to large numbers of MPI ranks on the Cray XC architecture. Together the solution enables users to:

- Perform complex simulations involving the interaction of multiple physics efficiently
- Simulate bigger, more detailed models and more complex physics
- Deliver more innovation, more features and more reliability in less time
- Deliver higher productivity, better product quality and faster time-to-market



ANSYS CFX Simulation, 36.03 million cells
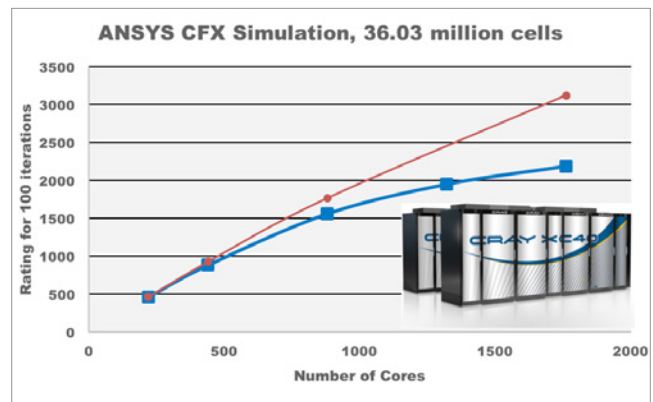
## ANSYS CFX and the Cray XC System in Action

The Benchmark Supercritical Wing test case was developed as a part of the Benchmark Models Program at NASA Langley Research Center. This program facilitated the development and validation of aeroelastic CFD codes by providing steady and unsteady flutter experimental data. The wing has a

TOGETHER, CRAY AND ANSYS ARE COMMITTED TO HELPING BRING AEROSPACE APPLICATIONS TO NEW HEIGHTS OF SIMULATION FIDELITY.
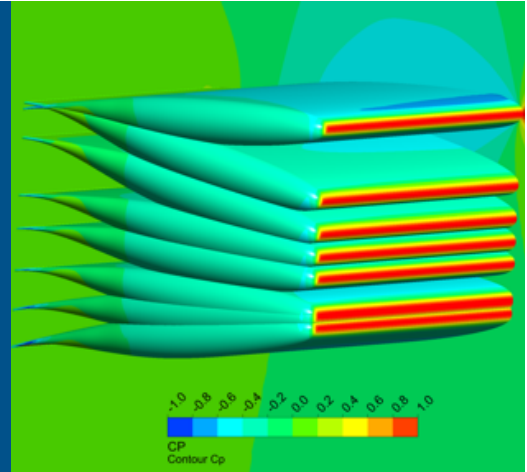


rectangular platform with a supercritical airfoil shape cross section.

For this case, researchers obtained unsteady experimental data by using pitch and plunge apparatus (PAPA) at the specified flutter dynamic pressure of 168.8 psf. This case is validated using the ANSYS CFX 6DOF solver on a Cray supercomputer. The hybrid Tet + Prism mesh with three different mesh refinements were created using ANSYS FLUENT meshing software (i.e., 1.75 million, 5.03 million and 13.47 million nodes) and performed both steady and dynamic case simulations using ANSYS CFX 17.0. Finally, the researchers studied the scalability of the ANSYS CFX solver using a Cray XC40 supercomputer for the steady case using the finest mesh (13.47 million nodes) with scaling up to 1,760 cores.

For ANSYS CFX, the Cray XC system offers excellent parallel performance, with continued scaling with more than 1,000 cores for this 13.47-million-node simulation.

### Results Comparison

The computationally obtained results were validated with experimentally obtained results from the NASA Langley Transonic Dynamics Tunnel using flexible-mount PAPA, which provides two degrees-of-freedom wing flutter dynamic motion.

The frequency response function of pressure coefficient magnitude at 95 percent wingspan shows excellent agreement with the experimental data. The
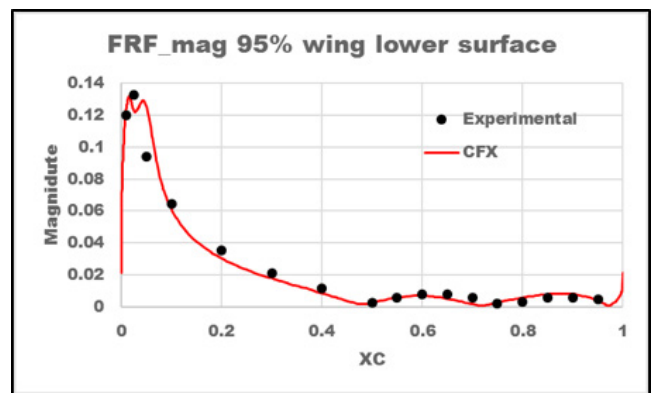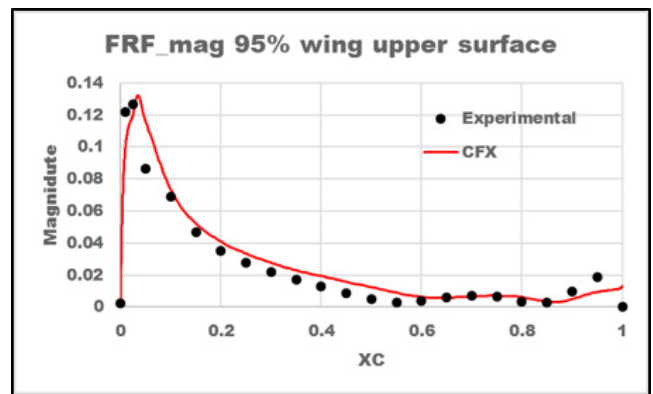




magnitude trends over the upper and lower wing surfaces are accurately captured and also match well with the experiments.

The results of this project clearly show that the accurate prediction of complex aeroelastic wing flutter benefits from the performance of the Cray XC system.