



## **Ansys + Leonardo Helicopters**

“Our ability to harness the power of Ansys Fluent native multi-GPU solver has saved us considerable time while using a fraction of the typical hardware resources.”

— **Francesco Manara**

Numerical Aerodynamics Technical Leader / Leonardo Helicopters

## **/ Faster Modeling with Fewer Resources: Leonardo Helicopters and Ansys Fluent**

Helicopters use three aerodynamic forces to fly: lift, thrust, and drag. Fine-tuning the design of a helicopter relies on precise engineering, analysis, and the harmony of all contributing factors.

Leonardo Helicopters has implemented Ansys simulation to help provide its customers with advanced rotorcraft that perform at the highest safety standards. Aerodynamics play a critical role in a helicopter's ability to lift off, navigate, and land safely and accurately. Achieving reliable flight requires extensive research and development. Thanks to Ansys computational fluid dynamics (CFD) software, Leonardo can reach aerodynamic optimization in record time.

### **/ Challenges**

Helicopters go through extensive certification stages that include both performance and safety standards. Improper evaluation of a rotor design, for example, can lead to aerodynamic problems and jeopardize safety.

Therefore, when modeling, it is imperative that the entire airframe and all its external parts are included. Ansys Fluent® fluid simulation software helps Leonardo optimize the research and development of the helicopter's aerodynamics for the rotorcraft.

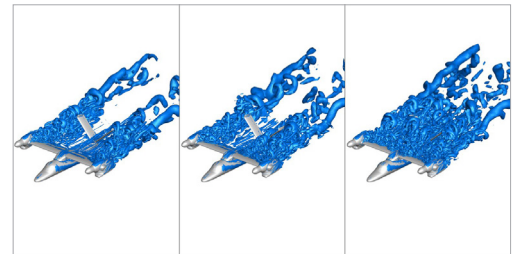
Additionally, testing use cases in the real world is extremely costly and not environmentally sustainable. For example, running a test in which a helicopter flies very close to a mountainside would cost thousands of dollars and generate up to 850 kg of CO2 emissions for just one hour of flight.

### **/ Technology Used**

- Ansys Fluent® fluid simulation software

### **/ Engineering Solutions**

Leonardo Helicopters relied on Ansys computational fluid dynamics (CFD) software powered by GPUs to accelerate the aerodynamic analysis vital for safe operation. To improve reaction time to customer requests, Leonardo used Fluent's multi-GPU solver to compress development time and test changes quicker than ever. The GPU solver also provided the ability to run large eddy simulations (LES) for the purpose of testing mathematical models for turbulence in CFD.



*CFD visualizations of turbulent structures on a tiltrotor in airplane mode*



*AW139 search and rescue cabin crew in flight*



*Passenger transport AW139 performing offshore operations*

## / Benefits

- Models were run 2.6 times faster using only one-third of the hardware resources when compared to previous solvers.
- Ansys simulation yielded faster results with fewer computational resources.
- An estimated energy reduction from 85 kWh to 15 kWh resulted in an 80% reduction in energy use.

**Ansys Fluent CFD software powered by GPUs enabled the models to run 2.6 times faster, resulting in an 80% reduction in energy use.**

## / Company Description

Leonardo is a global industrial group that builds technological capabilities in aerospace, defense, and security. The company plays a prominent role in major international strategic programs and is a trusted technological partner of governments, defense agencies, institutions, and enterprises.

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724-746-3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

When visionary companies need to know how their world-changing ideas will perform, they close the gap between design and reality with Ansys simulation. For more than 50 years, Ansys software has enabled innovators across industries to push boundaries by using the predictive power of simulation. From sustainable transportation to advanced semiconductors, from satellite systems to life-saving medical devices, the next great leaps in human advancement will be powered by Ansys.

*Ansys and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.*

Visit [www.ansys.com](http://www.ansys.com) for more information.

©2025 ANSYS, Inc. All rights reserved.