CFD-BASED RECONSTRUCTION OF IN-FLIGHT AERODYNAMIC LOADS ON A DASSAULT FALCON 8X

Malik Cheikh*, Vincent Fleury, Nicolas Forestier, Steven Kleinveld, Eric Garrigues, Aurélien Merlet

*malik.cheikh@dassault-aviation.com
Dassault Aviation,
78 quai Marcel Dassault – Cedex 300, 92552 Saint-Cloud,
France

ABSTRACT

As an aircraft manufacturer, Dassault Aviation extensively employs aerodynamic and aeroelastic modelling tools to optimize and certify the aerostructural design of its products. The in-house FEM tool ELFINI [1] and CFD code AETHER [2] were developed to meet this objective and have been consolidated for decades from a wide range of tests, including wind tunnel and in-flight aeroelastic tests, static tests and ground vibration tests. With the aim of achieving even greater precision, a finer assessment of the aero-structural computational processes is still needed to achieve higher performances. In the present paper, a detailed study aiming at gathering an extensive in-flight measurement database and reproducing by simulations several configurations on the wing of a Falcon 8X is proposed.

In June 2024, a flight test campaign was conducted on the Falcon 8X S/N 1 development aircraft at Dassault Aviation's Flight Test Centre in Istres, France. The in-flight instrumentation consisted of:

- 27 parietal pressure sensor pads (totalling around 600 sensors) located at five span stations on the left wing, on the winglet and on the lower side of the spoiler.
- A digital image correlation (DIC) system with four cabin-mounted cameras to track 94 targets, enabling wing shape measurements with a challenging precision of ±0.1° on the twist angle.

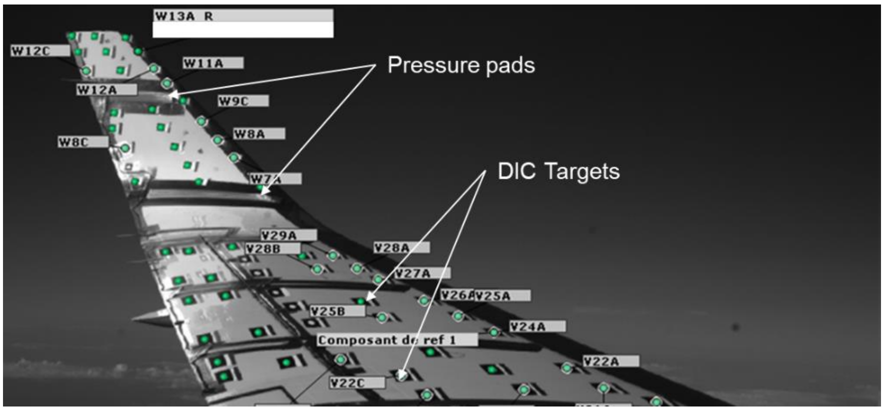

Figure 1 : Wing instrumentation

This paper presents the test campaign, which comprises three flights (totalling 16 hours 30 minutes) divided into 230 test sequences covering a broad portion of the Falcon 8X flight envelope (from Mach 0.3 to Mach 0.9 and slightly higher). Measurements were taken under various conditions, including steady level flight, steady turns, symmetric aileron deflections (±1°, ±2°, ±4°, ±10°), airbrake deployment, high-lift device operation, and some dynamic manoeuvres.

The paper also describes the computational flowchart to reproduce the measurements of steady pressure over the wing. The first step of the approach consists in deforming the reference wing geometry to match the distribution of the twist angles measured along the spanwise direction of the wing. A 3D mesh is then generated accordingly and CFD computations are carried-out taking into account the actual aerodynamic and engine flight conditions. The static pressure coefficient over the wing is finally compared to the measurements for validation. For computational efficiency purpose, the 3D mesh is straightforwardly derived from a reference non-twisted 3D mesh and the whole process performs automatically, from the modification of the wing geometry to the CFD results.

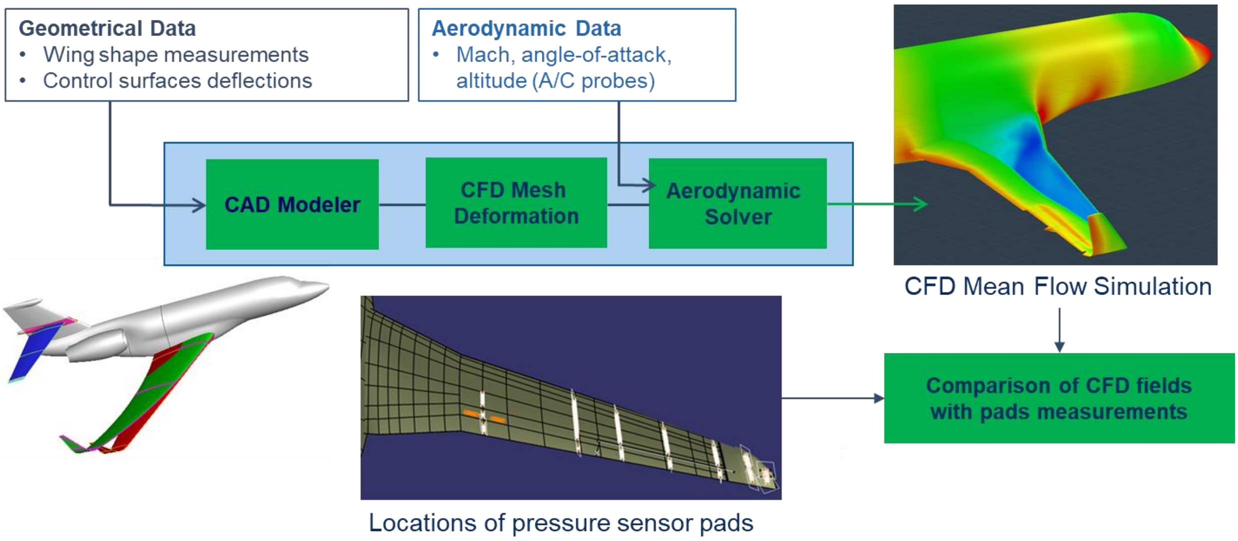

Figure 2 : Computational flow chart for CFD reconstruction of in-flight aerodynamic loads

The final paper will address an analysis of the flight tests. Briefly, the correlation of the CFD data with the flight measurements is generally satisfactory. Deviations are observed in the transonic regime near the aileron region. In this region, the shock is predicted too far downstream by simulation, while the aerodynamic load over the aileron is accurately reproduced though. Comparison between computations with rigid aileron deflections and ones taking into account its aeroelastic deformation consolidates design assumptions. Simulations with spoiler also show promising results in presence of strongly separated flow. Illustrations and technical details will be presented in the final paper.

A research program co-funded by the Direction Générale de l'Aviation Civile (DGAC) has supported this work.

References

[1] Garrigues, E. (2018). A review of industrial aeroelasticity practices at Dassault Aviation for military aircraft and business jets. Aerospace Lab, 1-34.

[2] Chalot, F., Johan, Z., Mallet, M., Billard, F., Martin, L., and Barré, S. (2023). Extension of Methods Based on the Stabilized Finite Element Formulation for the Solution of the Navier–Stokes Equations and Application to Aerodynamic Design. Computer Methods in Applied Mechanics and Engineering, Vol. 417, 2023.