X International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2023
M. Papadrakakis, B. Schrefler and E. Oñate (Eds)

AN HPC MULTI-PHYSICS FRAMEWORK FOR NEXT-GENERATION INDUSTRIAL AIRCRAFT SIMULATIONS

Ettore Fadiga*, **Francesco Rondina***, **Stefano Oliani***, **Tommaso Benacchio**[#†], **Daniele Malacrida*** and **Luigi Capone***

*Leonardo Labs
Leonardo S.p.A.
Via Pieragostini, 80, Genova, Italy
e-mail: {ettore.fadiga.ext, francesco.rondina.ext, stefano.oliani.ext, daniele.malacrida.ext, luigi.capone}@leonardo.com

#Leonardo Labs, Future Rotorcraft Technologies
Leonardo S.p.A.
Via G. Agusta 520, Samarate(VA), Italy

†Danish Meteorological Institute
Lyngbyvej 100, 2100 Copenhagen, Denmark
e-mail: tbo@dmi.dk

Key words: Fluid dynamics, Computational Mechanics, Exascale Computing, Engineering, Industrial Applications.

Abstract. The aerodynamic performance of industrial aircraft is strongly coupled with the structural deformation under aerodynamic load. Consequently, multi-physics simulations represent an important asset during the design and optimization phases. Most of the Fluid-Structure Interaction (FSI) coupling approaches adopted by researchers and engineers can be divided into monolithic and partitioned techniques.

The monolithic coupling, which consists of a unique system of equations for the coupled problem, is usually characterized by higher robustness and easier scalability. However, this approach is inherently customized for specific applications and requires a significant development effort. The partitioned coupling links existing software on a higher level, benefitting from higher flexibility and lower time-to-solution. As the computational world marches towards the exascale, black-box coupling libraries such as preCICE [2] aim to combine the flexibility and user-friendliness of partitioned approaches with complete and efficient usage of the available computational power.

This paper focuses on aeroelasticity analyses currently performed at the Leonardo Labs facilities, exploiting the recently installed davinci-1 supercomputer [14]. Open-source CFD and structural dynamics software applications are coupled using preCICE to conduct fully three-dimensional FSI analyses of aeroelastic test cases of industrial interest. The activities are part of a broader Digital Innovation industrial strategy centred on Digital Twins combining high-fidelity, highly scalable numerical simulations with data-driven AI models.

1 INTRODUCTION

Aircraft structures exposed to aerodynamics forces incur in flutter instabilities, caused by positive feedback between the deformation of the wings and the force exerted by the air. One of the most common engineering practice involves the employment of linear flutter analysis methods. As an example, doublet lattice flutter codes offer reliable predictions at subsonic Mach numbers, playing a central role in aircraft design. However, linear codes are not suitable for the prediction of flutter behavior of wings at transonic flow conditions, which characterize typical cruising speeds of large commercial and military transport aircraft [1]. Additionally, supersonic transport aircraft and fighters are often operating near Mach 1, where the flutter margin minimum is typically located due to non-linear flutter phenomena.

When the flow field locally exceeds the critical Mach number, shock waves appear on the surface of wings. In some cases, these shocks may induce flow separations. These phenomena have a strong effect on the aeroelastic behavior of aircrafts, since an immediate reduction of the flutter stability limit can be triggered. This sudden decrease of the flutter boundary is generally known as transonic dip [8].

The Aeroelastic Prediction Workshops (AePWs), organized by NASA in the context of AIAA forums, are focused on the investigation of aeroelastic effects with numerical tools that represent the current state of the art. In particular, the High Angle Working Group [13] has been employing coupled CFD/CSD simulations to study transonic flutter boundaries on the Benchmark SuperCritical Wing (BSCW) geometry [5]. The BSCW geometry was adopted as main object of the present work, and the related experimental data, released in the context of the AePW2 and AePW3 workshops, were used as validation references.

Computational Fluid Dynamics/Computational Structural Dynamics (CFD/CSD) coupled simulations are one the most accurate tools that engineers can employ to predict dynamic flutter in transonic conditions, characterized by highly non-linear effects. The goal of this work is to analyze and understand transonic flutter phenomena, evaluating the applicability of open-source tools for Fluid Structure Interaction (FSI) coupled simulations of transonic flutter effects.

Most of the FSI coupling approaches adopted by researchers and engineers can be divided into monolithic and partitioned techniques. The monolithic coupling, which consists of a unique system of equations for the coupled problem, is usually characterized by higher robustness and easier scalability. However, this approach is inherently customized for specific applications and requires a significant development effort. The partitioned coupling links existing software on a higher level, benefitting from higher flexibility and lower time-to-solution. As available computing resources cross the exascale, black-box coupling libraries such as preCICE [2] aim to combine the flexibility and user-friendliness of partitioned approaches with complete and efficient usage of the available computational power. The flexibility of a partitioned approach is extremely valuable in an industrial context, where different open-source and commercial software are employed on a daily basis. In this work, the OpenFOAM CFD software was coupled to the Finite Element Method (FEM) software CalculiX using the preCICE library functionalities.

The FEM software Calculix was chosen to solve the structural elasticity problems included in this article. The software is an open-source implementation that resembles the input structure of the widely used licensed software ABAQUS (R) and supports all the conventional linear and non-linear schemes. The software supports implicit static and dynamic solvers, explicit dynamic solvers, eigenvalue extraction, and is designed to run natively on Unix systems. In particular, the preCICE adapter for Calculix [12] historically supports the dynamic solver, activated via the *DYNAMIC keyword. However, recently, it has been extended to also support time-dependent modal analyses via the *MODAL DYNAMIC keyword.

The fluid dynamics side of the coupling was tackled by using the OpenFOAM CFD framework. In particular, the recently published density based implicit coupled solver dbnsFoam was adopted as fluid dynamic solver. This solver, which is part of the ICSFoam library developed by Oliani et al. [10], was connected to preCICE using the official OpenFOAM adapter developed by Chourdakis et al. [3].

The rest of the paper is structured as follows. Section 2 describes the methodology of the work, including details on the test case and experimental references. the CFD, CSD and coupled models. Simulation results are contained in Section 3, starting with uncoupled CFD, both steady state and transient, then moving on to the coupled CFD/CSD runs. Section 4 concludes the study with a discussion and concluding remarks.

2 METHODOLOGY

2.1 Test case and experimental references

The BSCW model is constructed from the NASA SC(2)-0414 airfoil and consists of a 0.4064 x 0.8128 m rectangular wing planform. It was tested twice in NASA laboratories to obtain both the flutter boundary (TDT Pitch and Plunge Apparatus - PAPA) [6] and the aerodynamics response to forced pitch oscillations of the wing (Oscillating Turntable - OTT) [11]. In both cases, the supercritical wing was mounted to a splitter plate, as represented in figure 1.

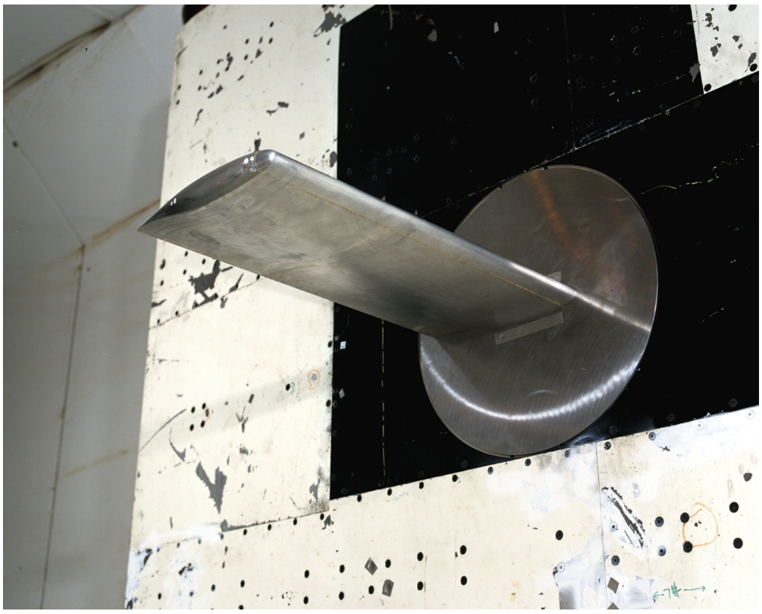

Figure 1: Benchmark SuperCritical Wing (BSCW) with splitter plate. [13]

For what concerns the PAPA configuration, the natural frequencies of the overall system were measured at 3.33 Hz for the plunge mode and 5.20 Hz for the pitch mode. The 60 % span location was populated with pressure transducers for both the experiments, while only for the PAPA tests the pressure measurements were additionally placed at the 95 % span location.

In this paper, a wide range of numerical simulations were carried out and validated with data from experimental campaigns. In particular, the authors conducted the following studies:

- **Pressure coefficient - steady state CFD**: the converged results of uncoupled CFD simulations were compared to temporal-averages of experimental findings obtained on the rigid static wing.
- **Forced oscillations - transient CFD**: the fluid-dynamics side of the coupling was validated in a stand-alone configuration against one of the OTT data-sets (Mach number of 0.7, Angle of Attack (AoA) of 3°).
- **Flutter boundary AePW2 - coupled CFD/CSD**: one of the configuration investigated in the second version of the AePW workshops was focused on an attached flow condition. In the spirit of gradually increasing the complexity of the analysis, the authors decided to initially test the coupled model with flow conditions of Mach 0.74 and AoA equal to 0 degrees.
- **Flutter boundary AePW3 - coupled CFD/CSD**: the third edition of the workshop considered a configuration characterized by an increased level of non-linearity. In particular, the last study presented in this work resembles the AePW3 conditions and was conducted at Mach 0.8 and AoA = 5°.

All the simulations were run with an input undisturbed dynamic pressure of 169 psf unless otherwise specified. In particular, different values of the dynamic pressure were used in the scope of flutter boundary simulations to identify the conditions correspondent to a zero-damping response.

2.2 CFD model

All the CFD simulations presented in this paper were performed with the density based implicit coupled solver **dbnsFoam**, included in the ICSFoam library [10]. The Reynolds-averaged Navier-Stokes (RANS) equations were computed with a backward scheme complete of pseudo time stepping for the temporal discretization, the $AUSM^+$ up convective flux scheme and the k-omega SST turbulence model [9].

The computational grids originate from a set of grids provided by NASA [4]. In particular, the coarse grid (3.6 million elements) and the medium grid (11.6 million elements) in wing-only configuration were employed in the present work. A freestream boundary condition was set at the external surfaces of the domain, except for the symmetry condition enforced at the symmetry plane.

Transonic flutter studies require a moving mesh treatment to accommodate the displacements of the wing surface. In this work, the authors used the **RBFMeshMotionSolver** developed in the scope of the solids4Foam project [15], where RBF stands for Radial Basis Function.

2.3 CSD model

The structural dynamics of the wing mounted on the PAPA system was modeled using the Finite Element Method (FEM) implemented in the CalculiX software. The mass properties, stiffness properties and pitch rotation axis of the original system were preserved while realizing a simplified model.

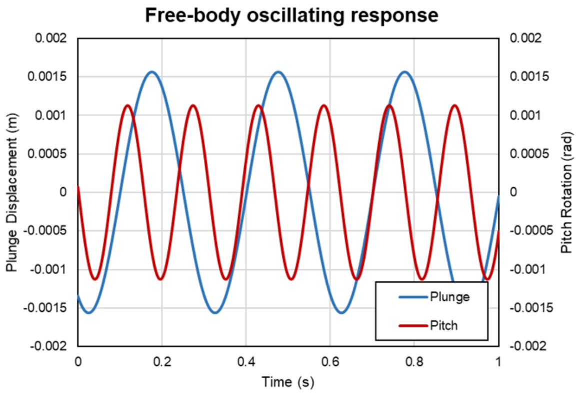

Figure 2: Free-body oscillating response of the FEM model, in terms of plunge displacement (blue) and pitch rotation (dark red) against time for the rigid body reference point.

The geometry of the wing consisted of 12k surface points connected to a reference point using the CalculiX rigid body feature. Due to the lack of torsional spring elements in CalculiX, the pitch rotation around the y axis is generated by means of two linear springs, respectively connected to additional points (included in the rigid body) and acting in the x direction.

Concentrated masses are defined in these two points to obtain the prescribed translational and rotational inertia of the PAPA structure. The plunge motion was accounted for with a spring acting on the z axis and connected to the reference point. Since the pitch and plunge springs act on different axes, the two modal motions are effectively decoupled. All the simulations were run using the explicit dynamic solver, an example of free-body oscillating response of the model is reported in figure 2

2.4 Coupled analyses with preCICE

For the FSI problem, the dbnsFoam simulation interfaces through preCICE with the Calculix model. In particular, an explicit coupling scheme was used throughout this work with a common timestep of 2.5e-4 s. The exchange of data at the interface (the shared surface between solid and fluid domains) consists in transferring the nodal displacement from the solid solver to the fluid solver, and nodal reaction forces in the opposite direction. In this particular case, the resolution of the FEM mesh at the interface does not influence the computational size of the problem, since the rigid body feature was used to construct the model. Consequently, it was possible to use a finely discretized interface with a nearest-neighbour mapping scheme.

The coupled simulations were started from a rigid steady CFD solution obtained with the standalone execution of the fluid solver. Then, the coupling with preCICE was activated, recording pitch rotations and plunge displacements for post processing operations.

3 RESULTS

3.1 Pressure coefficient - steady state CFD

The first validation test conducted by the authors regards the comparison between experimental time-averaged values of the pressure coefficient and the steady CFD solutions.

This validation test was run for all the configurations tested in the scope of this work. However, only the most challenging condition is included in the current paper with a comparison between the trends at 60 % of the span employing both the medium and the coarse grid. The comparison was run using the boundary and initial conditions corresponding to the AePW3 case with Mach number of 0.8 and AoA of 5°.

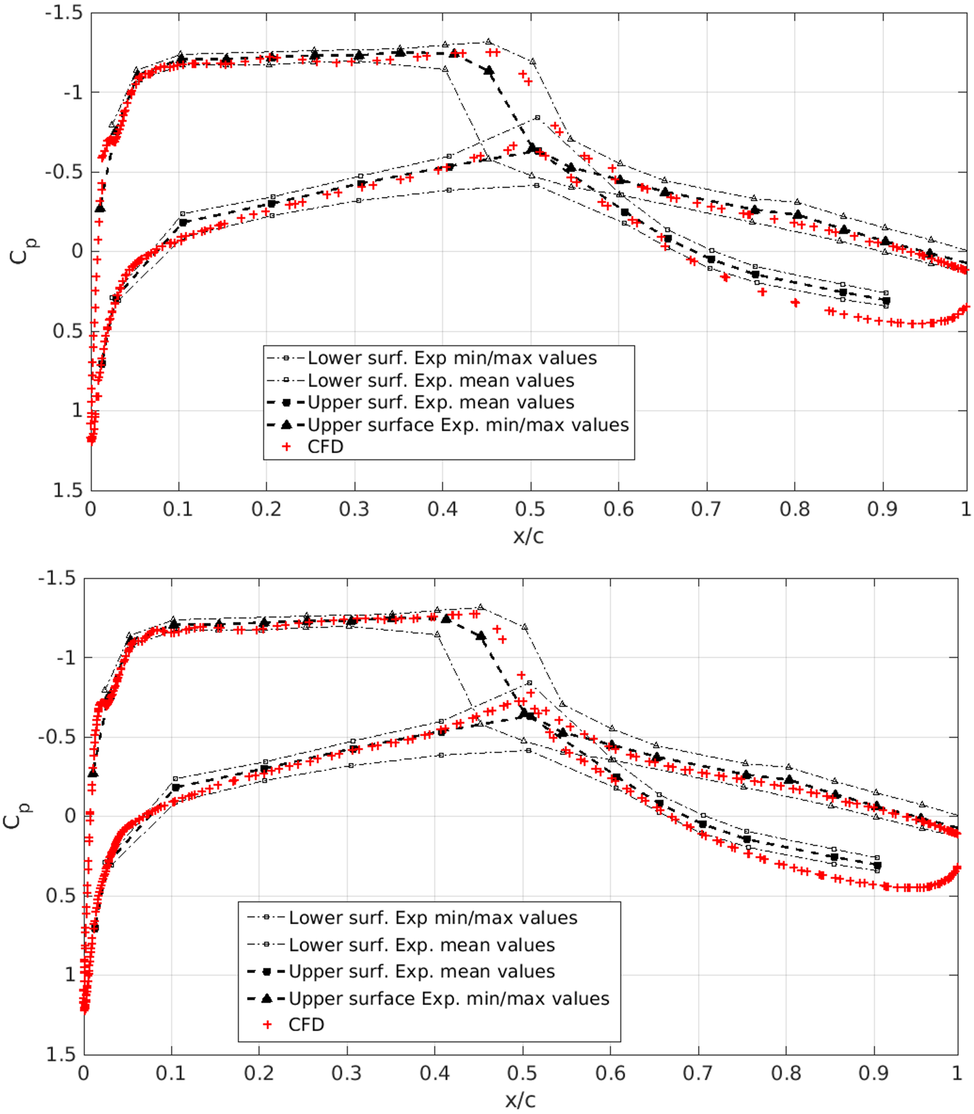

Figure 3: Pressure coefficient at 60 % span location - AePW3, 169 psf - top: coarse grid, bottom: medium grid.

The CFD results obtained with the coarse and medium grid are plotted in figure 3. The higher resolution of the medium grid in the near-shock region provides a better representation of the shock wave location. However, the coarse grid results already provide a close representation of the experimental data.

The analysis conducted on the AePW2 configuration (Ma 0.74, AoA 0°), characterized by a completely attached flow field, showed very good agreement between the CFD prediction and the experimental reference.

3.2 Forced oscillations - transient CFD

A wide range of forced oscillations studies was conducted by the participants to the second edition of the AePWs. The main objective was to define a critical pitch angle that represented a limit where the flow field was identified as significantly different from the one at lower angles of attack [7]. The critical angle was finally identified as $\alpha = 6.5$.

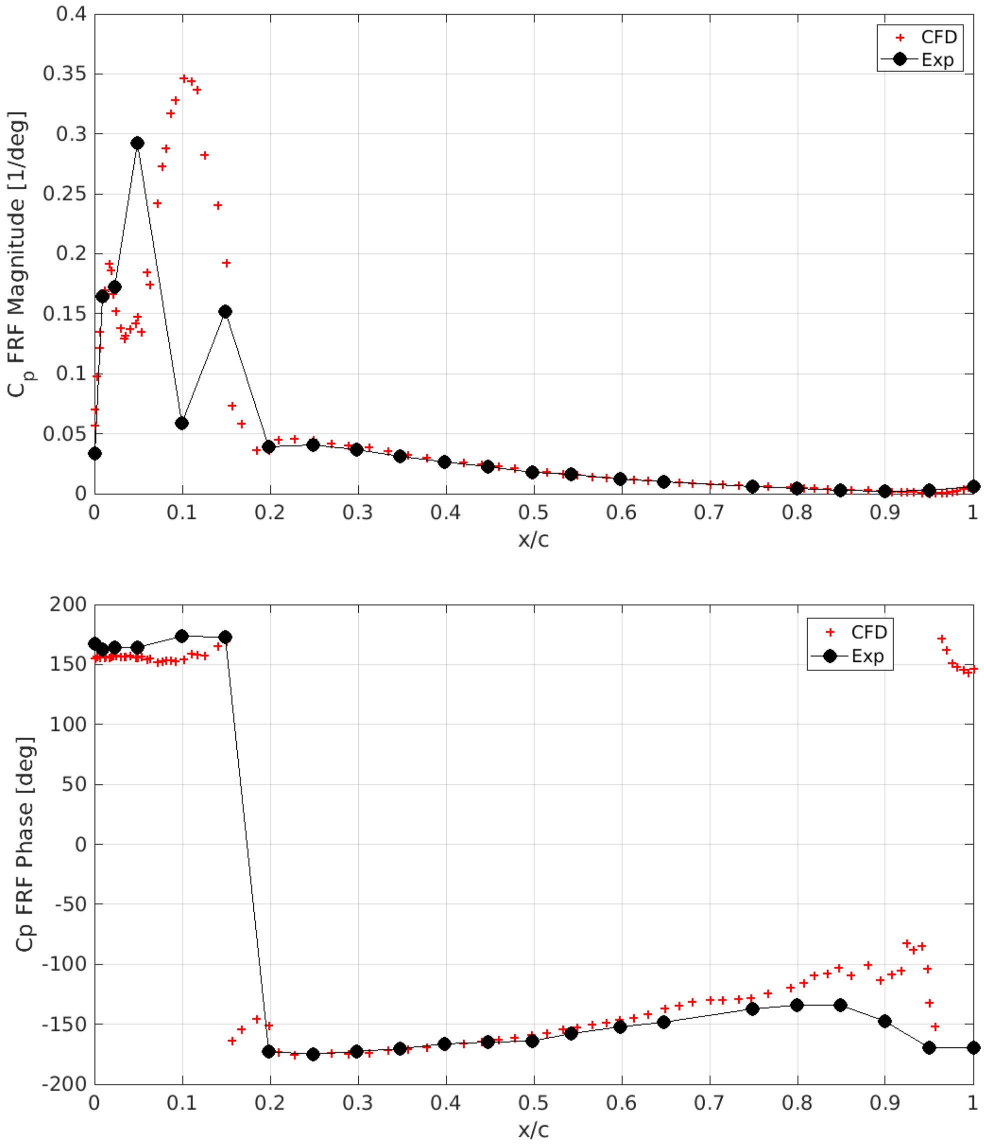

Figure 4: FRF magnitude and phase at 60 % span - upper surface - coarse grid.

In this study, the forced oscillations reference was not used to determine a critical angle. Instead, the experimental activities [11] characterized by 1 degree of oscillation amplitude were used to validate the dynamic mesh approach employed in the coupled analysis.

The results of the validation are reported in figure 4, where both the magnitude and the phase of the Frequency Response Function (FRF) at 60 % of the span are shown. Overall, computed data and experimental data are in good agreement, with differences in line with the general findings of workshop participants. The top right CFD phase values of figure 4 have been automatically translated of 360° during post processing operations. Considering the 2π periodicity, they also show a good agreement with the exerimental reference.

3.3 Flutter boundaries - coupled CFD/CSD

The identification of flutter boundaries was performed by running different CFD simulations with variable undisturbed dynamic pressure values. In particular, the coarse grid was used for both the AePW2 and AePW3 configuration analyses.

The experimental reference for the AePW3 case consisted of three different tests conducted at Mach 0.8 and slightly different AoA values (5.3°, 5.4° and 5.5°). The authors decided to run the current computations using an AoA of 5° to be aligned with the workshop. Since the critical measured flutter Qs were located between 90 and 125 psf, the presented case was structured with three different tests at 75, 100 and 169 psf.

The resulting pitch rotation trends are represented in figure 5, where the divergent behavior of the solution at 169 psf is easily appreciable. The other two conditions are closer to the flutter limit: quantitative analysis reported in figure 6 shows that also the 100 psf case has a divergent nature, while a positive damping ratio is recorded for the 75 psf conditions. Comparing the results with experimental values, the coarse grid seems to provide a slight underestimation of the flutter limit, showing a good agreement for what concerns the flutter frequency.

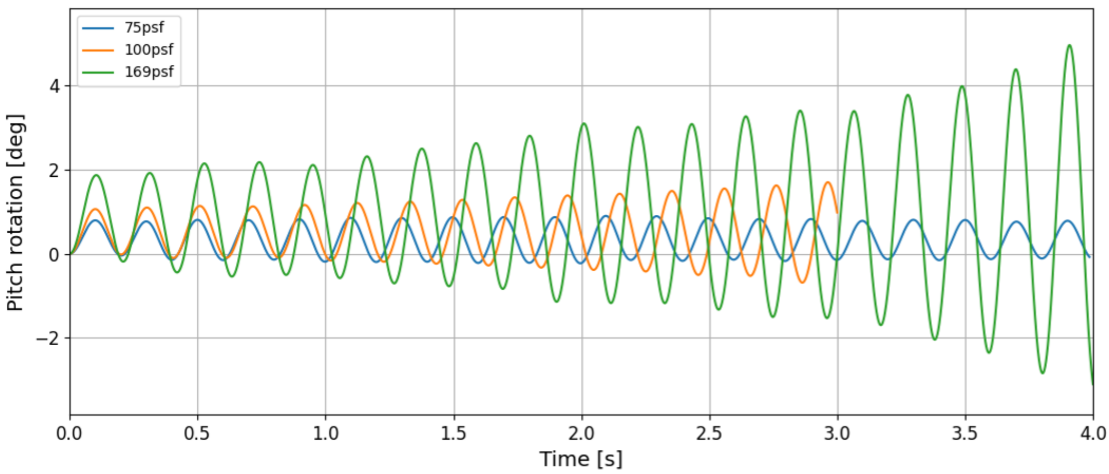

Figure 5: Pitch rotation - AePW3 - coarse grid

A similar study, reported in figure 7, was conducted for the AePW2 configuration. In this case, the experimental flutter boundary was 169 psf, value slightly higher than the CFD/CSD prediction located closer to 160 psf. However, the overall agreement between experimental

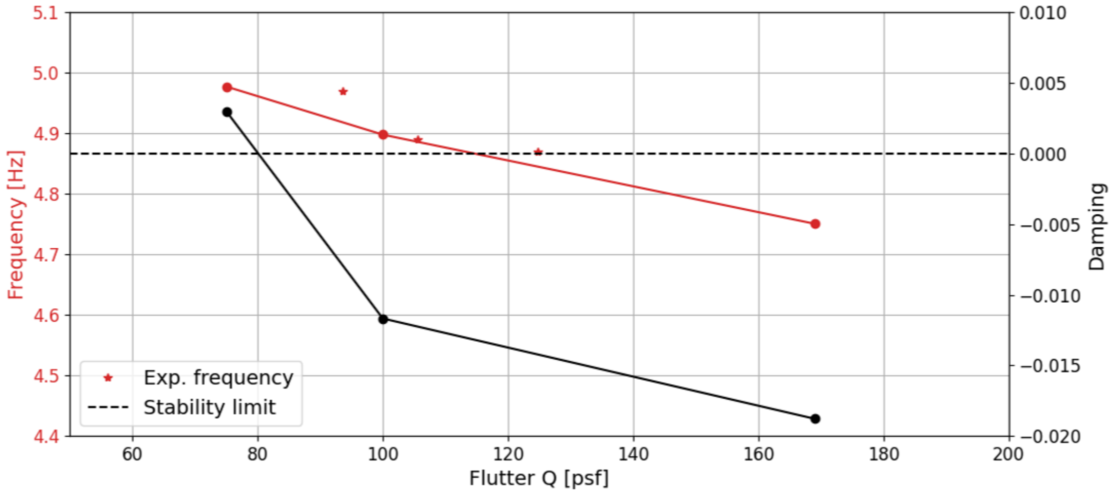

Figure 6: Flutter boundary and frequency - AePW3 - coarse grid

values and predicted frequency and flutter limit is considered satisfactory by the authors at this stage.

All the coarse grid simulations were conducted on 4 davinci-1 [14] CPU nodes, each characterized by 2 Intel(R) Xeon(R) Platinum 8260 CPU resulting in 48 total cores. In practice, using an overall value of 192 cores (191 for the CFD solver and 1 for the FEM solver) allowed to simulate 1 second in approximately 11 computational hours.

On the other hand, employing the medium grid on 8 nodes - 384 cores, led to a computational time of approximately 24 hours. Considering the relevant computational effort linked with the medium grid, the authors decided to focus on a single configuration (100 psf, Mach 0.8, AoA 5°) to evaluate the effect of mesh refinement on the flutter solution. The pitch rotation computed using the two grids is reported in figure 8. The flutter prediction shows a high sensitivity to the mesh refinement level, considering that the medium grid does not produce a divergent behavior at this dynamic pressure input.

4 DISCUSSION & CONCLUSIONS

A coupled simulation framework for aeroelastic studies was set up and tested on the modern HPC facility davinci-1 [14]. This framework, consisting of OpenFOAM, preCICE and CalculiX, is characterized by proved stability and can be upgraded to include new software via creation of new preCICE adapters.

The coupled system was tested on a state of the art aeroelastic test case regarding the analysis of transonic flutter, developed in the context of NASA AePWs [13]. The tests showed that a coupled CFD/CSD methodology is able to correctly predict flutter boundaries in highly non-linear transonic conditions using the workshop coarse grid. At the same time, the predicted solution appeared to be particularly sensitive to the fluid dynamic mesh resolution: the employment of the medium grid had a significant effect on the predicted flutter limit. Furthermore, investigating the effects of other aspects of CFD modeling (e.g., turbulence) would certainly drive additional interesting considerations.

The high fidelity results obtained with the coupled simulations framework are associated

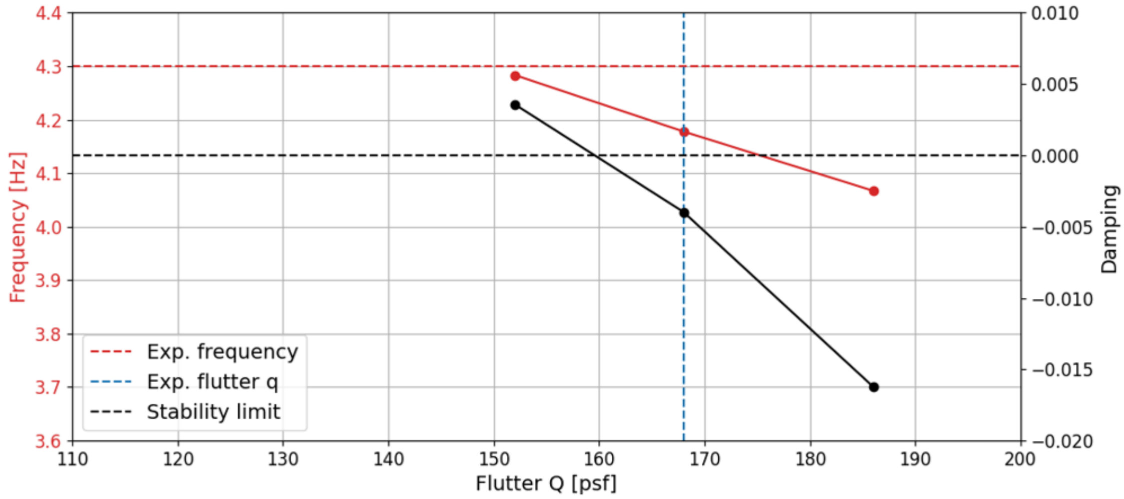

Figure 7: Flutter boundary and frequency - AePW2 - coarse grid

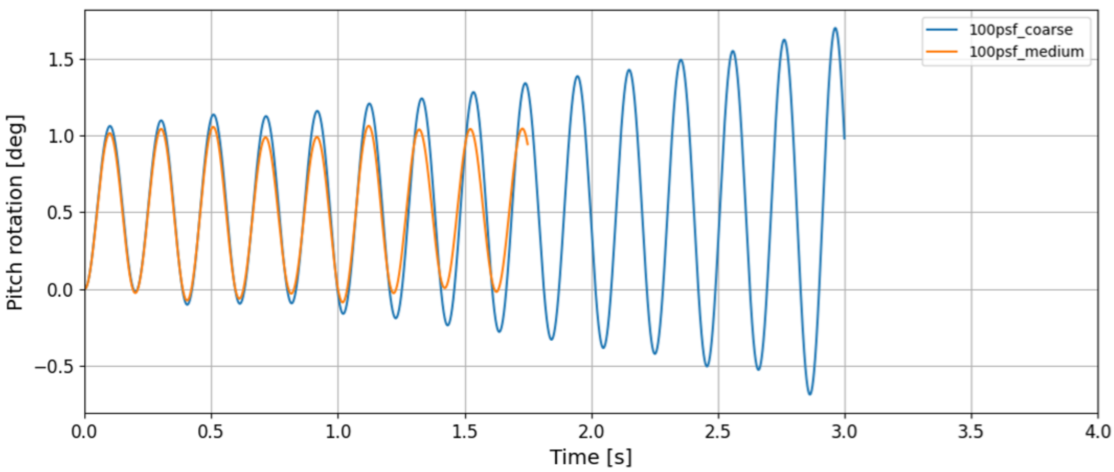

Figure 8: Pitch rotation 100 psf - AePW3 - coarse grid vs medium grid

with a considerable usage of computational resources (wall clock time of 24 hours for 1 second of simulation with the medium grid). Considering that at least three simulations are required to evaluate a single flutter boundary on a specific configuration (fixed Mach number and AoA), the CFD/CSD coupled method is inherently characterized by a relevant time-to-solution.

In particular, the RBF-based mesh deformation has a significant impact on the computational performance of coupled simulations. However, in the authors' experience, OpenFOAM built-in Laplacian methods tend to decrease the mesh quality when prismatic layers are employed and to introduce mesh-related instabilities for this kind of numerical simulations. Research efforts in these fields could increase significantly the industrial perspective of fully coupled simulations using OpenFOAM and preCICE.

In general, identifying the boundaries of a cost-effective usage of this methodology would be one important result of future research activities. Traditional flutter methods produce satisfying results in subsonic and attached flow conditions. However, off-design conditions characterized by high-angle configurations and transonic Mach numbers introduce non-linearities that require more accurate techniques. The seamless integration of traditional methods and coupled simulations would be an important goal to improve aeroelastic design workflows in industrial environments.

5 ACKNOWLEDGEMENTS

The authors would like to acknowledge Ivan Spisso for his useful insights on the OpenFOAM software suite and the preCICE development team for their valuable support in the initial stages of this project.

6 CRediT AUTHOR STATEMENT

E Fadiga: conceptualization, methodology, software, validation, formal analysis, writing - original draft, writing - review & editing, visualization, project administration. **F. Rondina:** conceptualization, methodology, software, formal analysis, writing - review & editing, visualization. **S. Oliani:** conceptualization, methodology, software, validation, formal analysis, writing - review & editing, visualization. **T. Benacchio:** conceptualization, writing - review & editing, supervision, project administration. **D. Malacrida:** validation, formal analysis, writing - review & editing. **L.Capone:** conceptualization, project administration, supervision, funding acquisition.

REFERENCES

[1] Bendiksen O.O. *Review of unsteady transonic aerodynamics: Theory and applications.* Progress in Aerospace Sciences 47 (2)(2011) 135-167

[2] Chourdakis G., Davis K., Rednberg G. et al. *preCICE v2: A sustainable and user-friendly coupling library [version 2; peer review: 2 approved].* Open Res Europe 2022, 2:51

[3] Chourdakis G., Schneider D., and Uekermann B. (2023). *OpenFOAM-preCICE: Coupling OpenFOAM with External Solvers for Multi-Physics Simulations.* OpenFOAM® Journal, 3, 1–25.

[4] Chwalowski P., Heeg J., and Biedron R. (2017). *Numerical investigations of the benchmark supercritical wing in transonic flow.* AIAA Paper 2017-0190.

[5] Dansberry, B.E. *Dynamic Characteristics of a Benchmark Models Program Supercritical Wing.* AIAA 92-2368, Apr. 1992.

[6] Dansberry B., Durham M., Bennett R., et al. *Experimental unsteady pressures at flutter on the supercritical wing benchmark model.* AIAA paper 93-1592-CP. Presented at the AIAA Structural Dynamics and Materials Conference, April 1993.

[7] Heeg J., Chwalowski P. *Predicting Transonic Flutter Using Nonlinear Computational Simulations.* International Forum on Aeroelasticity and Structural Dynamics (IFASD 2019)

[8] Isogai K., *On the transonic-dip mechanism of flutter of a sweptback wing.* AIAA Journal, Vol. 17, No. 7, 1979, pp. 793-795.

[9] Menter F., Carregal Ferreira J., Esch T., Konno B., and AC Germany. "The sst turbulence model with improved wall treatment for heat transfer predictions in gas turbines." In Proceedings of the international gas turbine congress, pages 2–7, 2003.

[10] Oliani S., Casari N., Carnevale M. *ICSFoam: An OpenFOAM library for implicit coupled simulations of high-speed flows.* Computer Physics Communications, Volume 286, 2023, 108673

[11] Piatak D. and Cleckner C. *Oscillating turntable for the measurement of unsteady aerodynamic phenomenon.* Journal of Aircraft. Vol 40, No. 1, Jan-Feb 2003.

[12] Uekermann B., Bungartz H.J., Cheung Yau L., Chourdakis G. and Rusch A. *Official preCICE Adapters for Standard Open-Source Solvers.* In Proceedings of the 7th GACM Colloquium on Computational Mechanics for Young Scientists from Academia, 2017.

[13] https://nescacademy.nasa.gov/workshops/AePW3/public/wg/highangle

[14] https://www.leonardo.com/en/innovation-technology/davinci-1

[15] https://solids4foam.github.io/